NASA Conference Publication 10003

Aeropropulsion '87

Session 3—
Internal Fluid Mechanics Research

Preprint for a conference at
NASA Lewis Research Center
Cleveland, Ohio, November 17-19, 1987

NASA

Aeropropulsion '87

Session 3—
Internal Fluid Mechanics Research

Preprint for a conference at
NASA Lewis Research Center
Cleveland, Ohio, November 17–19, 1987

NASA

National Aeronautics and
Space Administration

CONTENTS

PRECEDING PAGE BLANK NOT FILMED

INTERNAL FLUID MECHANICS

Brent A. Miller
and
Louis A. Povinelli

ABSTRACT

Internal Fluid Mechanics research at Lewis is directed toward an improved understanding of the important flow physics affecting aerospace propulsion systems, and applying this improved understanding to formulate accurate predictive codes. To this end, research is conducted involving detailed experimentation and analysis. The presentations to follow summarize ongoing work and indicate future emphasis in three major Research Thrusts, namely, Inlets, Ducts and Nozzles; Turbomachinery; and Chemical Reacting Flows.

Achievement of this goal requires that carefully planned and executed experiments be conducted in order to develop and validate useful codes. It is also important that the ultimate user community be part of the process in order to assure relevancy of the work and to hasten its practical application.
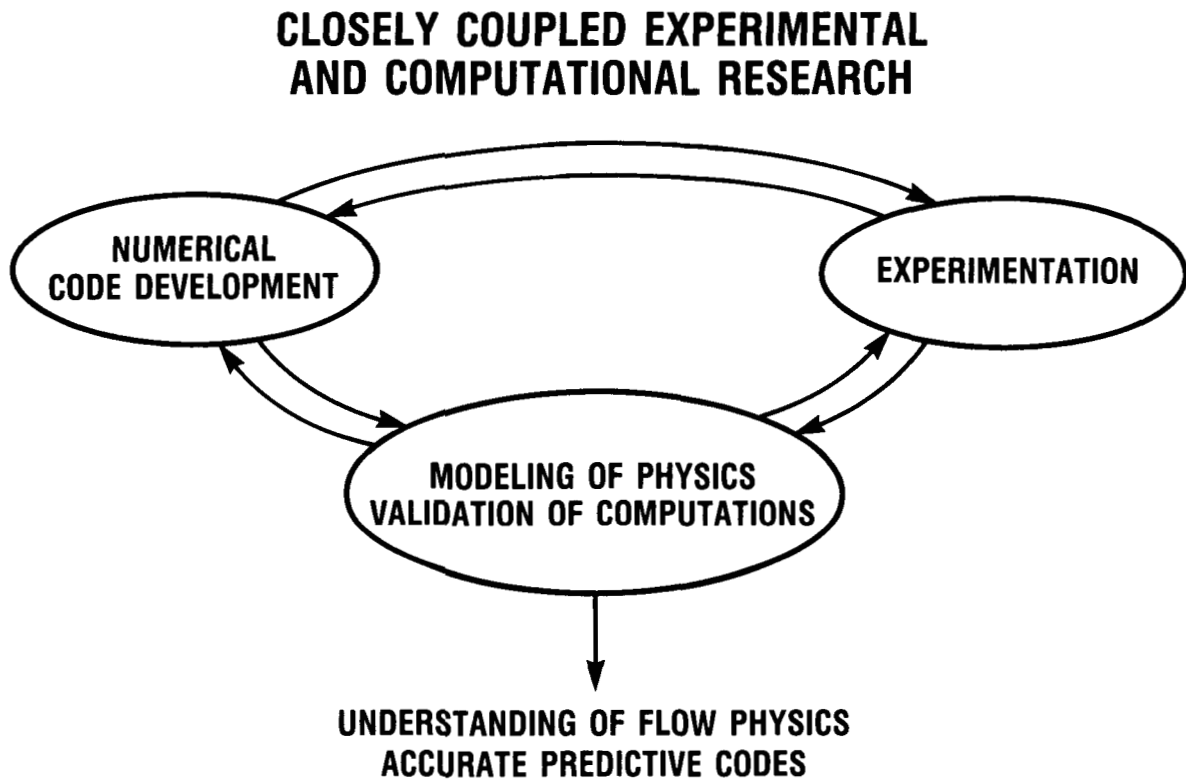
GOAL

TO BRING INTERNAL COMPUTATIONAL FLUID MECHANICS TO A STATE OF PRACTICAL APPLICATION FOR AEROSPACE PROPULSION SYSTEMS

CD-87-29476

It is critical that numerical code development work and experimental work be closely coupled. The insights gained are represented by mathematical models that form the basis for code development. The resultant codes are then tested by comparing them with appropriate experiments in order to ensure their validity and determine their applicable range.

CLOSELY COUPLED EXPERIMENTAL AND COMPUTATIONAL RESEARCH

NUMERICAL CODE DEVELOPMENT

EXPERIMENTATION

MODELING OF PHYSICS VALIDATION OF COMPUTATIONS

UNDERSTANDING OF FLOW PHYSICS
ACCURATE PREDICTIVE CODES

CD-87-29477

3-3

Propulsion systems are characterized by highly complex and dynamic internal flows. Many complex, three-dimensional flow phenomena may be present, including unsteadiness, shocks, and chemical reactions. By focusing on specific portions of a propulsion system, it is often possible to identify the dominant phenomena that must be understood and modeled for obtaining accurate predictive capability. In the following presentations on Internal Fluid Mechanics, research emphasis is placed on Inlets, Ducts and Nozzles, on Turbomachinery, and on Chemical Reacting Flows. These three Research Thrusts serve as a focus leading to greater understanding of the relevant physics and to an improvement in analytical tools. This in turn will hasten continued advancements in propulsion system performance and capability.

RESEARCH THRUSTS

HIGHLY 3-D FLOWS   UNSTEADY FLOWS   TURBULENCE/TRANSITION   SHEAR LAYERS/MIXING

SHOCK/BL INTERACTION   MULTIPHASE FLOWS   CHEMICAL KINETICS   COMBUSTION

RESEARCH THRUSTS

INLETS, DUCTS, AND NOZZLES   TURBOMACHINERY   CHEMICAL REACTING FLOWS
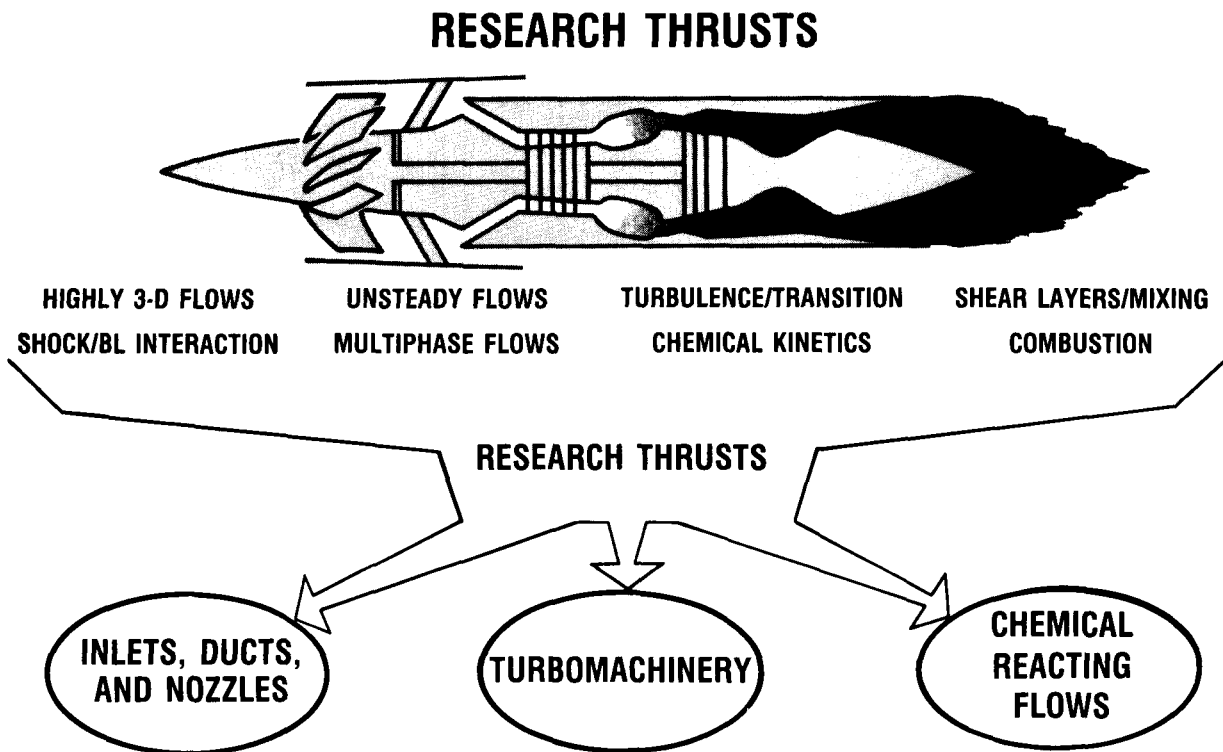
CD-87-29478

INLETS, DUCTS, AND NOZZLES

John M. Abbott,
Bernhard H. Anderson,
and
Edward J. Rice

ABSTRACT

The internal fluid mechanics research program in inlets, ducts, and nozzles is described. The program consists of a balanced effort between the development of computational tools (both parabolized Navier-Stokes and full Navier-Stokes) and the conduct of experimental research. The experiments are designed to better understand the fluid flow physics, to develop new or improved flow models, and to provide benchmark quality data sets for validation of the computational methods.
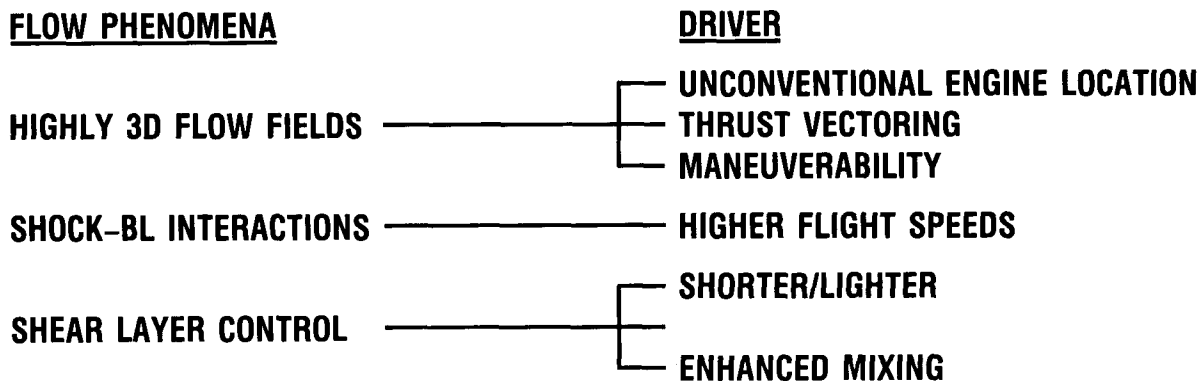
The inlet, duct, and nozzle research program is described according to three major classifications of flow phenomena: (1) highly three-dimensional flow fields, (2) shock - boundary-layer interactions, and (3) shear layer control. Specific examples of current and future elements of the research program are described for each of these phenomena. In particular, the highly three-dimensional flow field phenomena is highlighted by describing the computational and experimental research program in transition ducts having a round-to-rectangular area variation. In the case of shock - boundary-layer interactions, the specific details of research for normal shock - boundary-layer interactions are described. For shear layer control research in vortex generators and the use of aerodynamic excitation for enhancement of the jet mixing process are described.

Future research in inlets, ducts, and nozzles will include more emphasis on three-dimensional full Navier-Stokes methods and corresponding experiments designed to concentrate on the appropriate three-dimensional fluid flow physics.

The internal fluid mechanics research program in inlets, ducts, and nozzles is described according to three types of fluid flow phenomena – highly three–dimensional flow fields, shock – boundary-layer interactions, and shear layer control. The importance of each of these flow phenomena is a result of the drivers listed on the figure. For example, highly three–dimensional internal flow fields result from unconventional engine locations where twisting and turning inlets, ducts, and nozzles must be designed to deliver the airflow to and from the free stream. Aircraft thrust vectoring requirements quite often lead to the transitioning of nozzle cross-sectional geometries from round to rectangular with resultant three–dimensional flows. Aircraft maneuverability requirements can lead to significant three–dimensional flow fields entering the propulsion system inlet and ducting system. The push toward higher flight speeds, for both military and civilian applications, leads to the importance of research in shock – boundary-layer interactions within inlets, ducts, and nozzles. The desire to design inlets, ducts, and nozzles to be as short and light as possible points to the importance of shear layer control as a means for "stretching" the limits of the geometry while avoiding internal flow separations. Shear layer control in another sense, that is, the use of aerodynamic excitation to control the formation and development of a mixing layer, offers the potential for enhancing the mixing process in external nozzle flows.

INLETS, DUCTS, AND NOZZLES

FLOW PHENOMENA   **DRIVER**

HIGHLY 3D FLOW FIELDS ───────── ┌── UNCONVENTIONAL ENGINE LOCATION
                                ├── THRUST VECTORING
                                └── MANEUVERABILITY

SHOCK–BL INTERACTIONS ───────── HIGHER FLIGHT SPEEDS

SHEAR LAYER CONTROL ───────── ┌── SHORTER/LIGHTER
                               └── ENHANCED MIXING

CD-87-29854

3-6

Specific elements of the inlet, duct, and nozzle research program are listed for each of the three flow field phenomena. The four elements highlighted in the figure will be expanded upon in the remainder of the presentation. Specifically, highly three-dimensional flow fields will be illustrated by describing the transition duct research program. The shock – boundary-layer interaction phenomena will be illustrated with a description of the normal shock – boundary-layer interaction research program. Shear layer control research will be illustrated with two examples of current programs: vortex generator research and enhanced jet mixing research. The remaining four elements of the overall program, that is, offset ducts, oblique shock – boundary-layer interactions, glancing sidewall shock – boundary-layer interactions, and boundary-layer bleed, will not be described in this presentation, although they are also key elements of the overall program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - **TRANSITION DUCTS**
  - **OFFSET DUCTS**

- **SHOCK-BOUNDARY-LAYER INTERACTIONS**
  - **NORMAL SHOCK-BOUNDARY LAYER**
  - **OBLIQUE SHOCK-BOUNDARY LAYER**
  - **GLANCING SIDEWALL SHOCK-BOUNDARY LAYER**

- **SHEAR LAYER CONTROL**
  - **VORTEX GENERATORS**
  - **BOUNDARY-LAYER BLEED**
  - **ENHANCED JET MIXING**

CD-87-29855

3-7

The approach to the research program in inlets, ducts, and nozzles consists of a balance between computational and experimental research. Computationally, much of the emphasis until recently has been on the development and validation of parabolized Navier-Stokes methods. More recently and for the future, more emphasis is being placed on three-dimensional full Navier-Stokes methods.

Shown on the right-hand side of the figure are the elements of the experimental program in inlets, ducts, and nozzles. They are aligned with the respective computational method and point out the close linkage between the computational and experimental elements of the program. Note that each of the program elements listed in the previous figure under the three different flow phenomena also appear here as specific experiments.

INLETS, DUCTS, AND NOZZLES
APPROACH

COMPUTATIONAL METHODS

3D PARABOLIZED
NAVIER-STOKES (PNS)

3D NAVIER-STOKES (NS)

EXPERIMENTS

(1) TRANSITION DUCTS
(2) OFFSET DUCTS
(3) SHOCK–BOUNDARY-LAYER
    INTERACTION
(4) VORTEX GENERATORS
(5) BOUNDARY-LAYER BLEED

(6) EXPERIMENTS (1) TO (5)
(7) SEPARATION FLOW PHYSICS
(8) NONORTHOGONAL SURFACES
(9) JET MIXING

CD-87-29856

Highly three-dimensional flow field research will be illustrated by describing the transition duct research program.

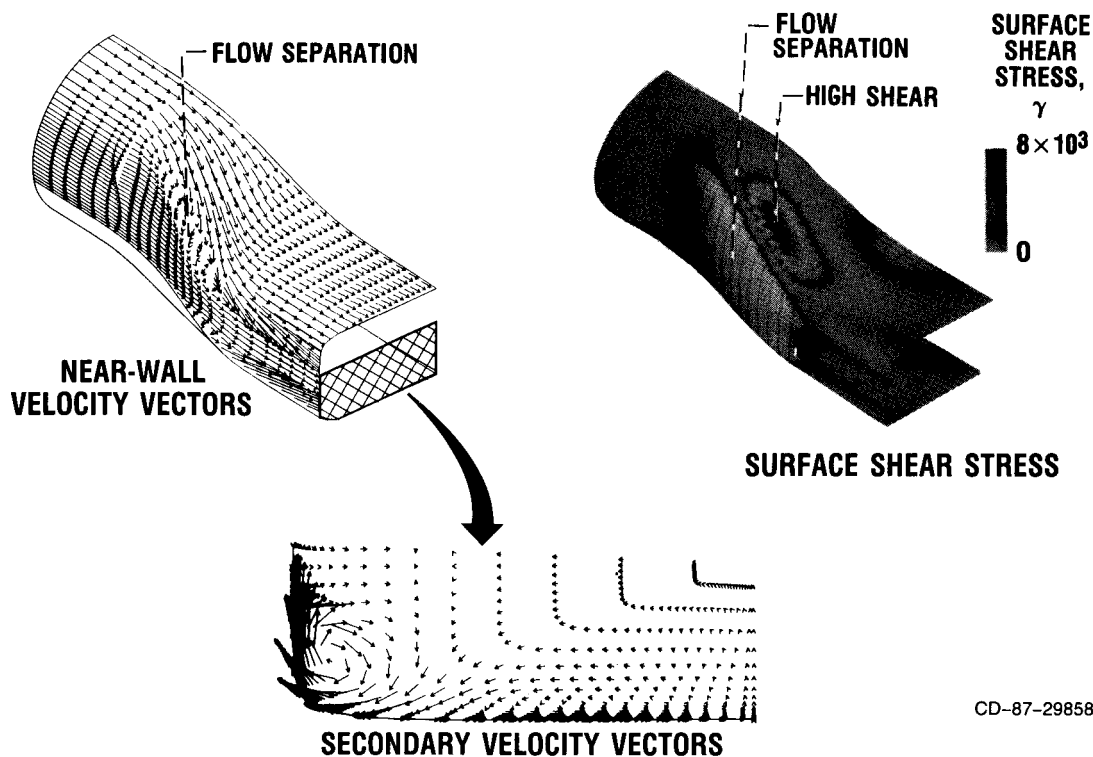# INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - **TRANSITION DUCTS**
  - **OFFSET DUCTS**

- **SHOCK–BOUNDARY-LAYER INTERACTIONS**
  - **NORMAL SHOCK–BOUNDARY LAYER**
  - **OBLIQUE SHOCK–BOUNDARY LAYER**
  - **GLANCING SIDEWALL SHOCK–BOUNDARY LAYER**

- **SHEAR LAYER CONTROL**
  - **VORTEX GENERATORS**
  - **BOUNDARY-LAYER BLEED**
  - **ENHANCED JET MIXING**

A sample three-dimensional parabolized Navier-Stokes (PNS) computation for a specific transition duct geometry is illustrated in this figure. The geometry is described by a rectangular exit having an aspect ratio (width/height) of 3.0. The duct is 3 inlet diameters long and has an exit-to-inlet area ratio of 1.0. The figure shows near-wall velocity vectors on the left with a display of the secondary velocity vectors in the rectangular exit plane shown below it. The three-dimensional character of the flow field is clear. On the right side of the figure are contours of surface shear stress on the duct. Zero shear stress corresponds to the onset of flow separation on the duct surface. Note that the computation indicates that this particular geometry has a small, localized separation zone about half way down the duct. The flow reattaches just downstream of this zone and remains attached for the remainder of the duct length. The separation is apparent in both the near-wall velocity vectors and in the surface shear stress contours.

TRANSITION DUCT, 3D-PNS COMPUTATIONAL RESULTS
## AR = 3.0, L/D = 1.5, A_{exit}/A_{inlet} = 1.0

— FLOW SEPARATION

NEAR-WALL
VELOCITY VECTORS

— FLOW
SEPARATION

— HIGH SHEAR

SURFACE
SHEAR
STRESS,
γ

8×10^3

0

SURFACE SHEAR STRESS

SECONDARY VELOCITY VECTORS

CD-87-29858

As an illustration of how the computation results shown in the previous figure can be used, this figure shows the internal flow separation bounds for a class of transition duct geometries. Each duct geometry has the same area ratio (1.0) but the length ratio and the exit aspect ratio are varied. Specifically, for four different aspect ratios, the length ratio was decreased from a value where the internal flow was completely attached to a value where the flow just began to separate. This series of computations then resulted in the separation bound shown in the figure. For geometries above the curve, the flow is attached, for geometries below the curve, the flow is separated. One experiment data point is spotted on the figure as a case where the flow was massively separated within the duct.

TRANSITION DUCT SEPARATION CHARACTERISTICS
### 3D-PNS COMPUTATIONS; $A_{exit}/A_{inlet}=1.0$
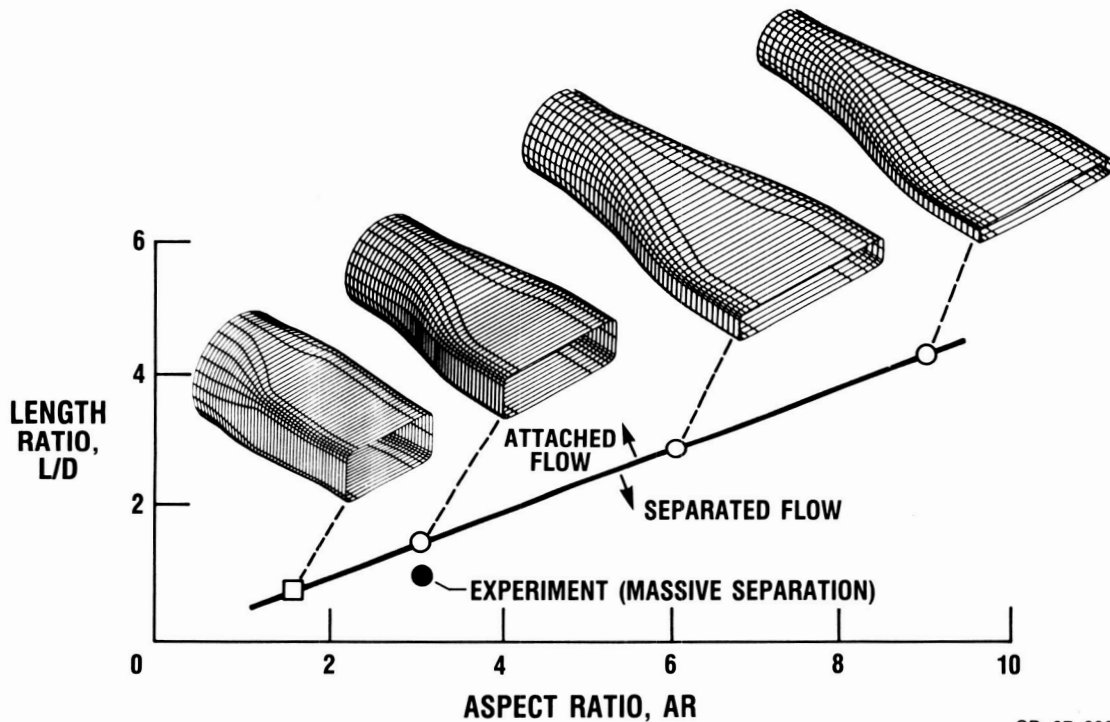
CD–87–29859

To develop a more detailed understanding of the flow physics within transition ducts, to improve models of the flow physics, and to validate computations like those shown in the previous two figures, experiments are underway using the model and facility shown in this figure. The model was machined to match exactly one of the geometries for which the three-dimensional PNS method had been applied. Shown in this figure is a photograph of the model during the final machining stage along with a superimposed display of the computational surface. Also shown is the test facility. Special care was taken to condition the transition duct inflow properly to provide the desired levels of inflow turbulence, flow angularity, and boundary-layer profile. Modifications to the facility are currently underway whereby the inflow tank is being connected to a high-pressure supply system to provide higher Reynolds number test capability.

TRANSITION DUCT AERODYNAMIC EXPERIMENTS
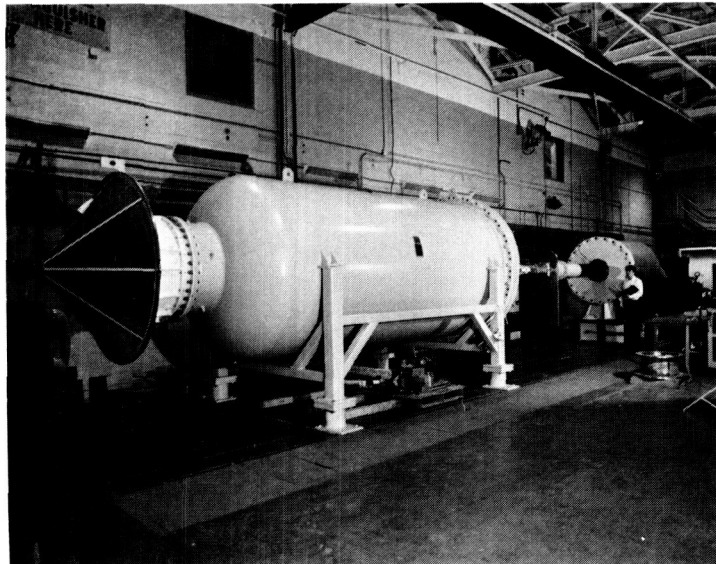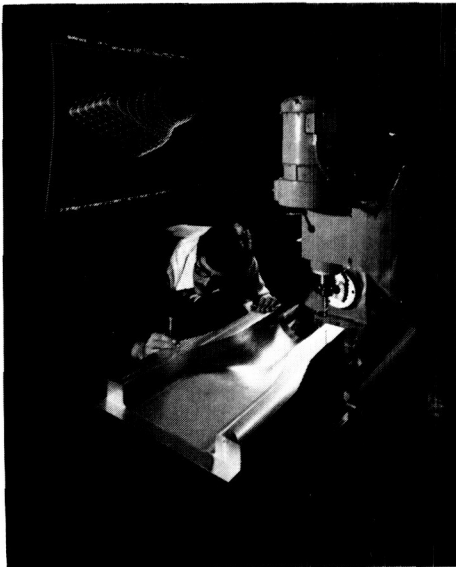
CD-87-29860
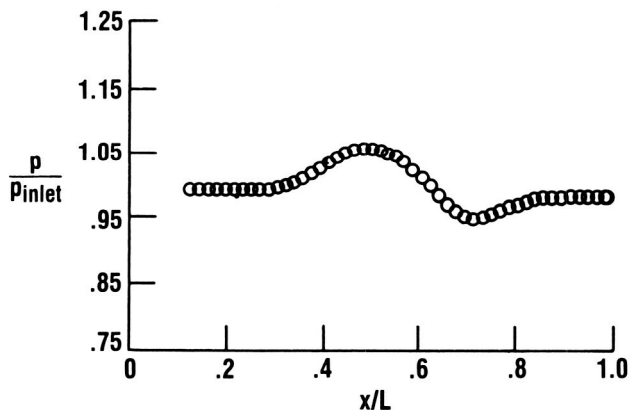
ORIGINAL PAGE IS
OF POOR QUALITY

Experimental results are shown in this figure at a one-dimensional duct Mach number of 0.5. On the left side of the figure, surface static-pressure measurements are shown along the centerline of the duct. On the right side of the figure are shown results from a surface-oil-streak flow-vizualization experiment. These oil streaks agree quite well qualitatively with the computational near wall velocity vectors. Although not shown here, experimental measurements have just been made of flow direction in the duct exit plane. Efforts are currently underway to compare those results with the corresponding three-dimensional PNS computations. Future experiments will include detailed probing of the flow field within this duct and others and direct measurement of surface shear stress using an advanced laser measurement technique.
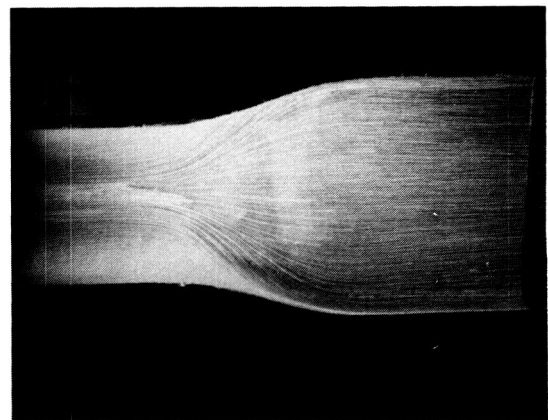
TRANSITION DUCT AERODYNAMIC RESULTS
MACH = 0.5

**CENTERLINE SURFACE STATIC PRESSURE DISTRIBUTION**

SURFACE OIL STREAKS

CD-87-29861

In addition to conducting aerodynamic experiments with the highly three-dimensional flow fields of transition ducts, heat-transfer experiments are also being conducted. Heat transfer is of particular concern for these types of flow fields because of applications where the three-dimensional flows may result in high-temperature flow streams (i.e., engine core flow) finding their way to the transition duct (nozzle) surfaces. The figure shows a square-to-rectangular transition duct model which was tested in the facility shown on the right. To measure the surface heat-transfer characteristics of the duct, the surface was coated with a liquid-crystal material. After establishing the desired amount of airflow through the duct, the duct surface was heated so that the resulting color bands (isotherms) on the liquid-crystal surface could be interpreted in terms of local heat-transfer coefficient.

TRANSITION DUCT HEAT TRANSFER EXPERIMENTS
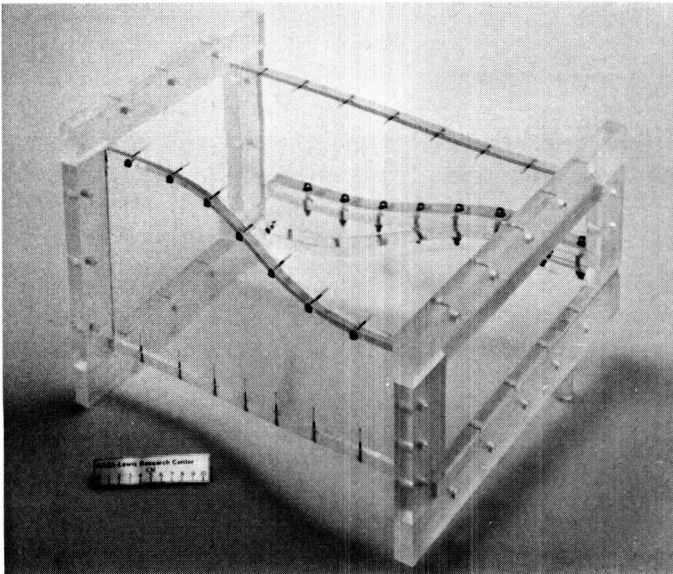
CD-87-29862

TRANSITION DUCT HEAT TRANSFER RESULTS

Results from the liquid-crystal heat-transfer experiment are illustrated in this figure. The different colors on the liquid-crystal surface correspond to different surface temperatures, which, when combined with the known level of surface heat input, lead to experimentally determined values of surface heat-transfer coefficient. Regions of high and low heat transfer are pointed out in the figure. By photographing the surface and then digitizing the photographic image, quantitative values of surface heat-transfer coefficient are obtained over the entire surface.

TRANSITION DUCT HEAT TRANSFER RESULTS
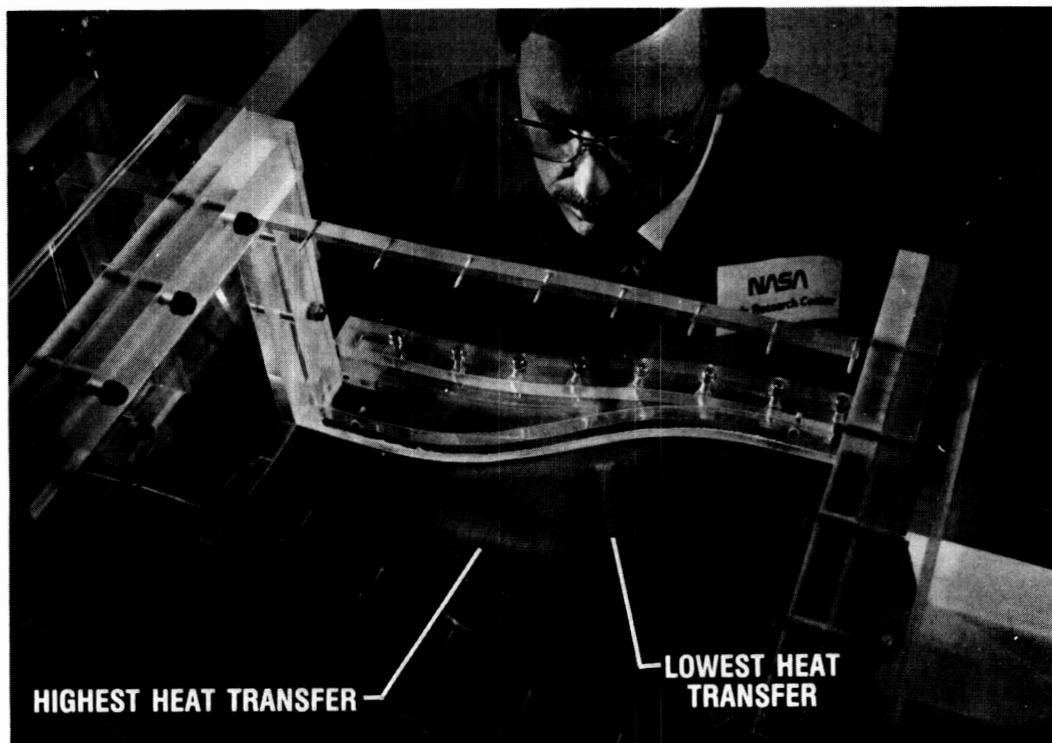
HIGHEST HEAT TRANSFER

LOWEST HEAT TRANSFER

CD-87-29863

Shock – boundary-layer interaction research will be illustrated by describing the normal shock – boundary-layer interaction research program.
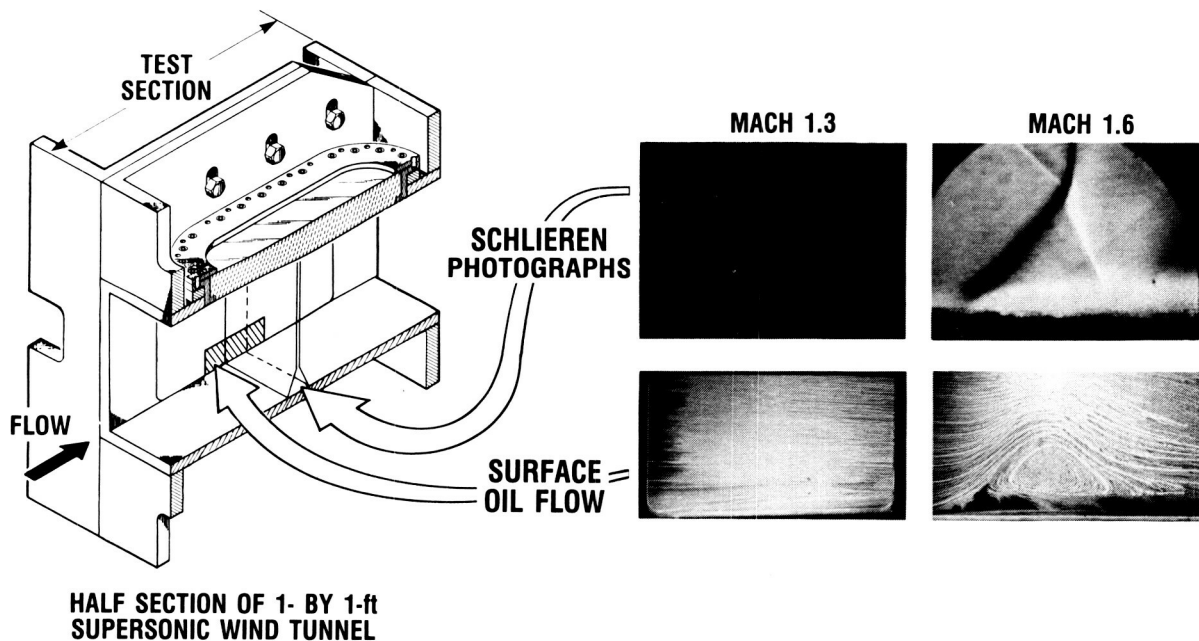
INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - **TRANSITION DUCTS**
  - **OFFSET DUCTS**

- **SHOCK-BOUNDARY-LAYER INTERACTIONS**
  - **NORMAL SHOCK-BOUNDARY LAYER**
  - **OBLIQUE SHOCK-BOUNDARY LAYER**
  - **GLANCING SIDEWALL SHOCK-BOUNDARY LAYER**

- **SHEAR LAYER CONTROL**
  - **VORTEX GENERATORS**
  - **BOUNDARY-LAYER BLEED**
  - **ENHANCED JET MIXING**

NORMAL SHOCK – BOUNDARY-LAYER INTERACTION

This figure illustrates the nature of the normal shock – boundary-layer interaction.
The interaction is shown at two Mach numbers, 1.3 and 1.6, and the nature of the
interaction is illustrated with a schlieren photograph and a surface oil streak
photograph at each Mach number. The photographs were taken in the Lewis 1- by 1-ft
supersonic wind tunnel. The schlieren photographs give a view of the interaction
integrated across the full width of the test section, while the surface oil streaks
illustrate the details of the flow field along the side wall of the test section.
At both Mach numbers the schlieren photographs show the shock being bifurcated into
a "lambda shock" in the interacting region. The sidewall surface oil
streaks indicate a significant difference in the structure of the flow field for the
two cases. At Mach 1.3 the flow along the sidewall remains uniform and passes
through the shock structure with no major alteration. At Mach 1.6 the adverse
pressure gradient across the shock is strong enough to force the boundary layer to
separate from the tunnel walls and to form the closed separation bubble shown in the
figure. Note that this is a two-dimensional slice through a highly three-dimensional
flow field that exists in the corner region.
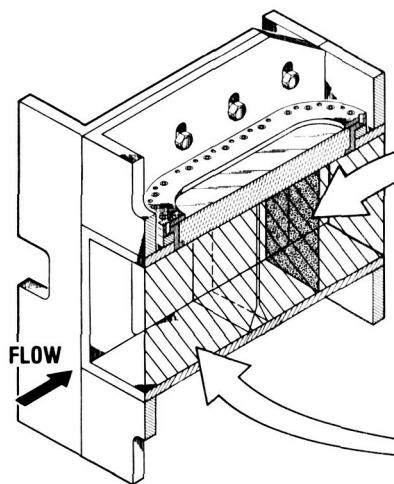
NORMAL SHOCK–BOUNDARY-LAYER INTERACTION

TEST SECTION

FLOW

SCHLIEREN PHOTOGRAPHS

SURFACE OIL FLOW

MACH 1.3

MACH 1.6

HALF SECTION OF 1- BY 1-ft
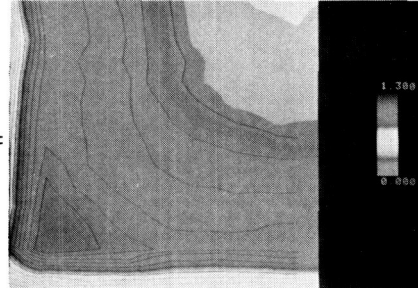SUPERSONIC WIND TUNNEL

CD-87-29864

Both the Mach 1.3 and 1.6 shock – boundary–layer interaction flow fields were surveyed in detail using nonintrusive laser anemometry. Results are shown for the Mach 1.6 case in two planes within the flow field. The top set of Mach contours illustrates the nature of the flow field in a cross plane downstream of the shock. The lower set of Mach contours shows the development of the flow field within a vertical plane passing through the centerline of the test section. The separated region in the vicinity of the initial interaction causes the actual flow area to contract downstream of the initial shock, leading to a reacceleration of the flow to supersonic Mach numbers. The flow then shocks down again, reaccelerates again due to the thickened boundary layer, and finally shocks down a final time.

RESULTS OF LASER ANEMOMETER MEASUREMENTS FOR NORMAL SHOCK-BOUNDARY LAYER INTERACTION AT MACH NUMBER 1.6
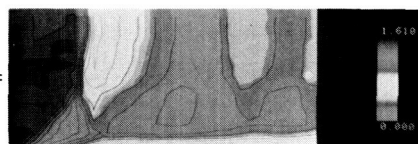

MACH NUMBER CONTOURS FROM MEASUREMENTS
MADE ON CROSS-PLANE DOWNSTREAM OF SHOCK

HALF SECTION SHOWING TYPICAL
MEASUREMENT PLANES

CD–87–29865

MACH NUMBER CONTOURS FROM MEASUREMENTS
MADE ON AXIAL MID-PLANE OF TUNNEL

NORMAL SHOCK – BOUNDARY-LAYER INTERACTION –
COMPARISON OF ANALYSIS AND EXPERIMENT

The lower portion of the previous figure is repeated as the lower portion of this
figure, that is, Mach contours measured with laser anemometry in a vertical plane
passing through the centerline of the test section at Mach 1.6. The upper portion
of the figure is a two-dimensional Navier–Stokes computation of the same flow field.
Although the computation captures fairly well the initial portions of the flow in
the vicinity of the first shock, in the downstream region, none of the flow physics
are adequately represented. This result is, of course, expected since the experi-
mental results have shown the strong three-dimensional character of the flow. This
comparison points out the need for three-dimensional computational methods for
computing such flow fields.

MACH 1.6 NORMAL SHOCK WAVE BOUNDARY LAYER INTERACTION
COMPARISON OF LDV EXPERIMENT AND ANALYSIS
ANALYSIS—2D NAVIER-STOKES

MACH NO.
—1.61
—0.0

FLOW
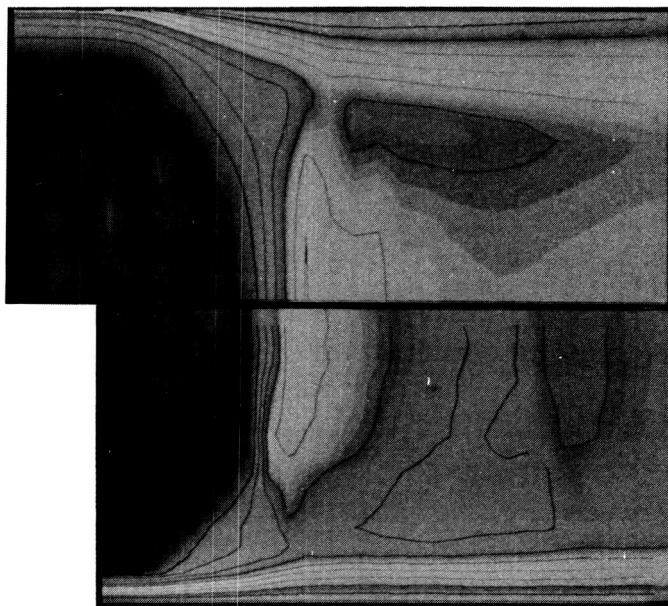
EXPERIMENT—LASER ANEMOMETRY

CD-87-29866

This figure is a frame from a film that will be shown during the conference. The film illustrates the additional insight and understanding that can be obtained from setting in to motion a three-dimensional contour representation of the flow field. The graphics software used to generate the film was actually developed for presenting computational results. The experimental laser anemometry data sets were modified to a format that was acceptable to the graphics software. The researcher can rotate the image, adjust the rate of rotation, and select the axis of rotation while sitting at the display console. One gains a perspective from these rotating images much more quickly than one would by looking at a series of two-dimensional or even three-dimensional plots.

NORMAL SHOCK–BOUNDARY-LAYER INTERACTION
LASER ANENOMETRY RESULTS PRESENTED WITH COMPUTATIONAL GRAPHICS PACKAGE
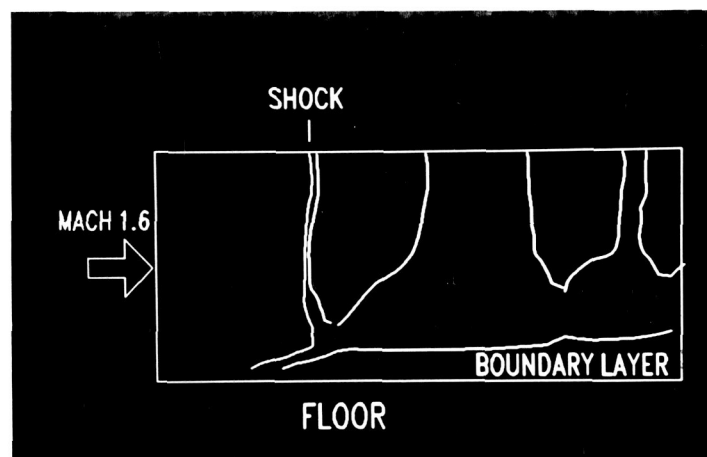
CD-87-29867

The first aspect of shear-layer control, boundary-layer control, will be illustrated by describing the vortex generator research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - **TRANSITION DUCTS**
  - **OFFSET DUCTS**

- **SHOCK–BOUNDARY-LAYER INTERACTIONS**
  - **NORMAL SHOCK–BOUNDARY LAYER**
  - **OBLIQUE SHOCK–BOUNDARY LAYER**
  - **GLANCING SIDEWALL SHOCK–BOUNDARY LAYER**

- **SHEAR LAYER CONTROL**
  - **VORTEX GENERATORS**
  - **BOUNDARY-LAYER BLEED**
 - **ENHANCED JET MIXING**

This figure illustrates a research model and facility that were used in an experiment to assess the performance of vortex generators in a diffusing offset duct. The duct had a length-to-diameter ratio of 5.0, an offset-to-diameter ratio of 1.34, and an exit-to-inlet area ratio of 1.50. Initial experiments with the duct identified the location of a separated flow region as shown by the surface oil streak photograph. Vortex generators were then added to the duct surface just upstream of the separated region to control the separation.
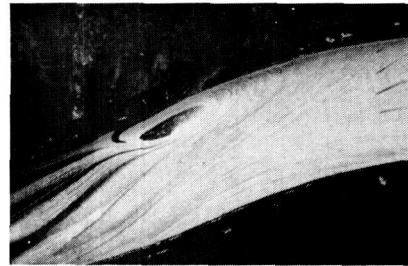
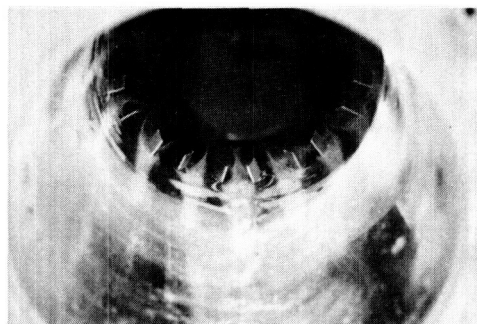## VORTEX GENERATORS IN DIFFUSING OFFSET DUCT
$L/D = 5.0$; $OFFSET/D = 1.34$; $A_{exit}/A_{inlet} = 1.50$
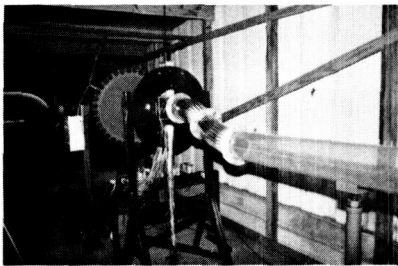
EXPERIMENT SETUP

**SURFACE OIL STREAKS
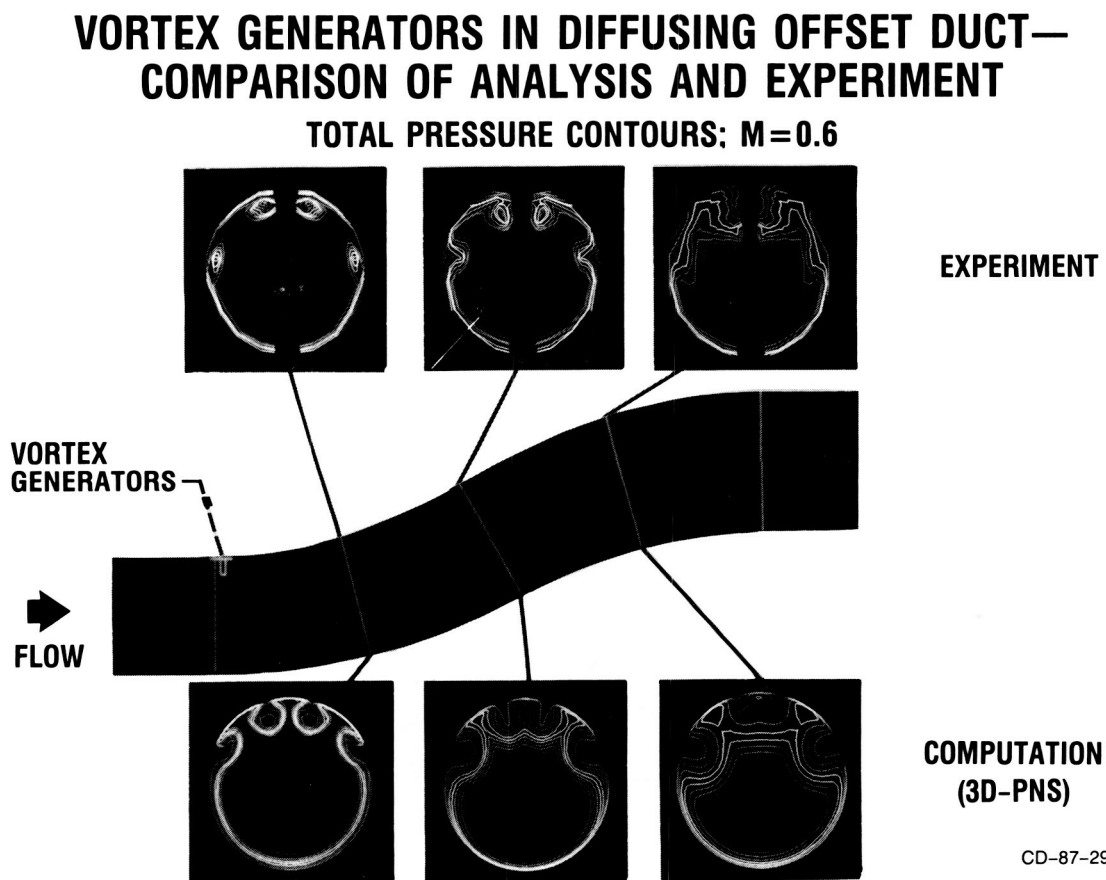WITHOUT VORTEX GENERATORS**

VORTEX GENERATORS

CD-87-29868

VORTEX GENERATORS IN OFFSET DUCT — COMPARISON OF ANALYSIS AND EXPERIMENT

A comparison of total-pressure contours at various locations down the length of the duct with the vortex generators in place, is shown in this figure. The experimental data were obtained by surveying the flow field with a total-pressure probe. The computational results are from the three-dimensional PNS method. The method includes a model for the vortex generators that allows one to position the individual generators anywhere within the duct and permits an adjustment to the strength of the vortex based on an empirical relationship with the vortex generator angle of attack.

In order to illustrate more clearly the influence of the vortex generators, a film will be shown of the total-pressure contours that one would see while moving downstream through the duct. Two segments of film will be shown — the first without the vortex generators and the second with the vortex generators.

VORTEX GENERATORS IN DIFFUSING OFFSET DUCT— COMPARISON OF ANALYSIS AND EXPERIMENT
TOTAL PRESSURE CONTOURS; M=0.6

EXPERIMENT

VORTEX GENERATORS

FLOW

COMPUTATION (3D-PNS)

CD-87-29869

3-23

Another aspect of the shear layer control, the use of aerodynamic excitation to control the development and evolution of large-scale structures in a shear layer, will be illustrated by describing the enhanced jet mixing research program.

INLETS, DUCTS, AND NOZZLES
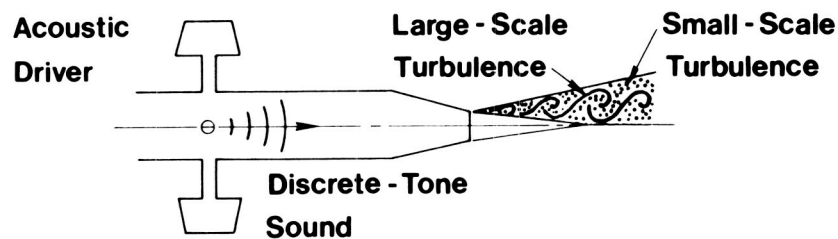## FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - **TRANSITION DUCTS**
  - **OFFSET DUCTS**

- **SHOCK–BOUNDARY-LAYER INTERACTIONS**
  - **NORMAL SHOCK–BOUNDARY LAYER**
  - **OBLIQUE SHOCK–BOUNDARY LAYER**
  - **GLANCING SIDEWALL SHOCK–BOUNDARY LAYER**

- **SHEAR LAYER CONTROL**
  - **VORTEX GENERATORS**
  - **BOUNDARY-LAYER BLEED**
  - **ENHANCED JET MIXING**

The mixing process between a jet and the surrounding air flow can be enhanced through
use of aerodynamic excitation. The naturally occurring flow structure of the jet
mixing process is shown in the left-hand Schlieren photograph. By applying an exci-
tation signal at the proper frequency, here by use of acoustic drivers upstream of
the jet exit plane, the naturally occurring large-scale structures within the mixing
layer are regularized and enhanced and lead to a more rapid mixing process as
illustrated in the center photograph.

ENHANCED JET MIXING

Coherent Large-Scale Turbulence in Jet Flows

Acoustic Driver **Large-Scale Turbulence** **Small-Scale Turbulence** **Discrete-Tone Sound**

Unexcited   Excited

CD-87-29870

Results of an axisymmetric Navier-Stokes computation of a jet flow exiting into a
quiescent region are shown in this figure. The jet exit Mach number is 0.3, and the
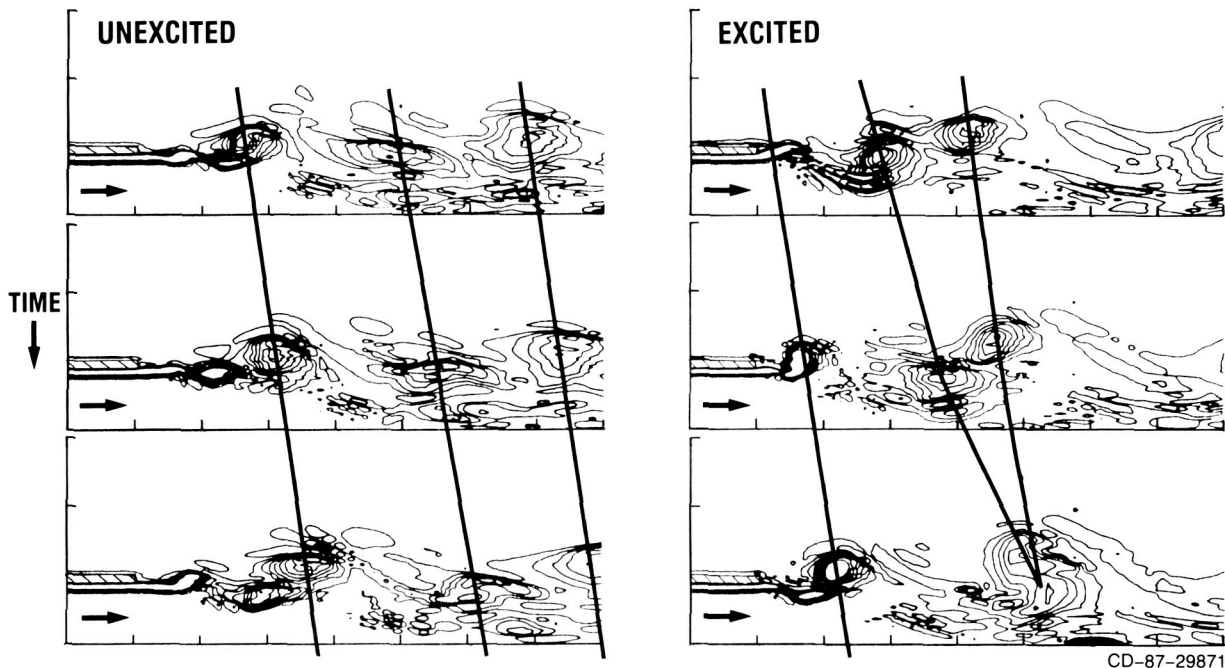results are shown in terms of vorticity contours. Two cases are shown – on the left
side an unexcited case and on the right side an excited case. For the excited case
the excitation signal is applied at a frequency chosen to maximize the development
and growth of the large-scale structures. In both cases results of the computation
are shown at three different times. Also for both cases lines are drawn through the
centers of the large-scale vortices to illustrate how they propagate downstream.
Note that, with excitation applied at the proper frequency, adjacent vortices combine
or pair to form a single-larger vortex, which, in turn, has the effect of more
rapidly mixing the jet flow with the surrounding quiescent flow.

EFFECT OF EXCITATION ON JET SHEAR LAYER EVOLUTION—
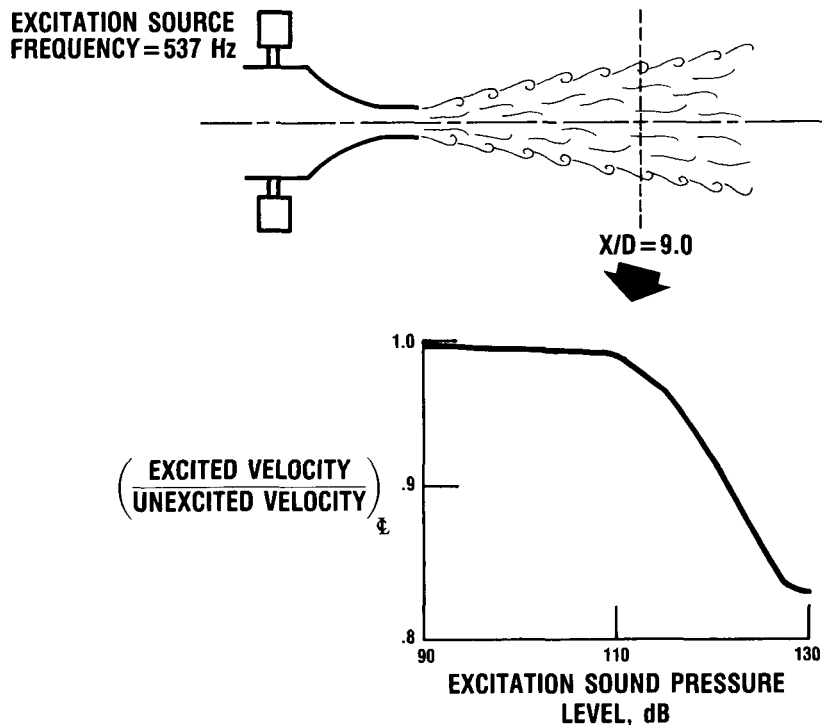# NAVIER-STOKES COMPUTATIONAL RESULTS
VORTICITY CONTOURS; MACH 0.3

CD-87-29871

The effect of excitation on the jet mixing process for a jet having an exit Mach number of 0.3 and an initial turbulence level of 0.15 percent is shown in this figure. The results were obtained in an experiment wherein both the frequency and the level of the excitation signal could be varied. The results show how the ratio of centerline velocity for the excited case to that of the unexcited case varies as the level of the excitation is increased at a fixed frequency. The measurements are made nine jet diameters downstream of the nozzle exit plane. As in the previous figure, the excitation frequency has been selected to provide maximum mixing enhancement. As indicated, the effect of the excitation is quite significant with, in this case, a reduction in centerline velocity of about 16 percent at the maximum available excitation level of 130 dB.

EFFECT OF EXCITATION ON JET MIXING—EXPERIMENTAL RESULTS
MACH=0.3; JET TURBULENCE=0.15%

EXCITATION SOURCE
FREQUENCY=537 Hz

X/D=9.0

$$\left(\frac{\text{EXCITED VELOCITY}}{\text{UNEXCITED VELOCITY}}\right)_{\mathbb{C}}$$

EXCITATION SOUND PRESSURE
LEVEL, dB

CD-87-29872

In summary, the internal fluid mechanics research program in inlets, ducts, and nozzles is a balanced effort between the development of computational tools and the conduct of experimental research. The program has been briefly described by high-lighting research efforts in highly three-dimensional flow fields, shock - boundary-layer interactions, and shear-layer control. Much of the computational focus until now has been on the development and validation of parabolized Navier-Stokes methods. More recently and in the future, more emphasis will be placed on the development and validation of three-dimensional Navier-Stokes methods. The experimental element of the program will continue to provide a fundamental understanding of the fluid flow physics, to develop new and/or improved flow models, and to provide benchmark data-sets for computational method validation of both the parabolized and full Navier-Stokes methods.

INLETS, DUCTS, AND NOZZLES
SUMMARY

COMPUTATIONAL METHODS

**3D PARABOLIZED
NAVIER-STOKES (PNS)**

EXPERIMENTS

**(1) TRANSITION DUCTS
(2) OFFSET DUCTS
(3) SHOCK–BOUNDARY-LAYER
    INTERACTION
(4) VORTEX GENERATORS
(5) BOUNDARY-LAYER BLEED**

3D NAVIER-STOKES (NS)	**(6) EXPERIMENTS (1) TO (5)** **(7) SEPARATION FLOW PHYSICS** **(8) NONORTHOGONAL SURFACES** **(9) JET MIXING**

CD-87-29873

TURBOMACHINERY

Robert J. Simoneau, Anthony J. Strazisar,
Peter M. Sockol, Lonnie Reid,
and John J. Adamczyk

ABSTRACT

The discipline research in turbomachinery, which is directed toward building the
tools we need to understand such a complex flow phenomenon, is based on the fact
that flow in turbomachinery is fundamentally unsteady or time dependent. Success in
building a reliable inventory of analytic and experimental tools will depend on how
we treat time and time-averages, as well as how we treat space and space-averages.

The raw tools at our disposal – both experimental and computational – are truly
powerful and their numbers are growing at a staggering pace. As a result of this
power, a case can be made that we are currently in a situation where information is
outstripping understanding. The challenge is to develop a set of computational and
experimental tools which genuinely increase our understanding of the fluid flow and
heat transfer in a turbomachine.

The following viewgraphs outline a philosophy based on working on a stairstep
hierarchy of mathematical and experimental complexity to build a system of tools,
which enable one to aggressively design the turbomachinery of the next century.
Examples of the types of computational and experimental tools under current
development at Lewis, with progress to date, are examined. The examples include
work in both the time-resolved and time-averaged domains.

Finally, an attempt is made to identify the proper place for Lewis in this continuum
of research.

The long-range goal of the turbomachinery research program at NASA Lewis is to establish a validated three-dimensional viscous analysis capability for multistage turbomachinery, including unsteady effects and surface heat transfer. A full range of experimental and computational tools are being brought to bear on this problem in order to genuinely increase our understanding of the fluid flow and heat transfer in a turbomachine. The key to this understanding is the selection and successful application of the right tools for the right job.

TURBOMACHINERY RESEARCH PROGRAM
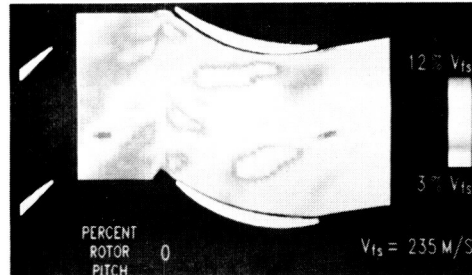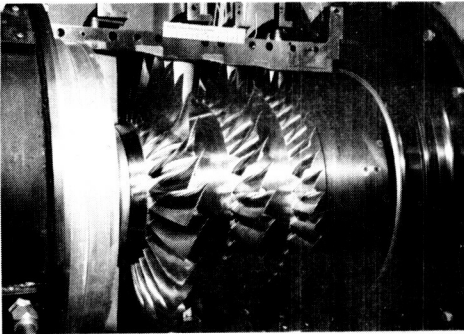
LONG-RANGE GOAL

ESTABLISH A VALIDATED 3D VISCOUS ANALYSIS CAPABILITY FOR MULTISTAGE TURBOMACHINERY, INCLUDING UNSTEADY EFFECTS AND SURFACE HEAT TRANSFER
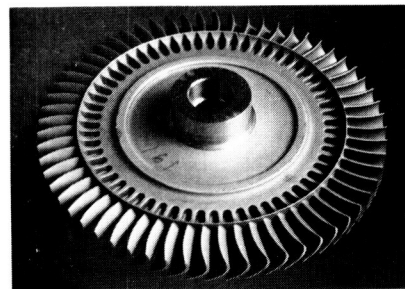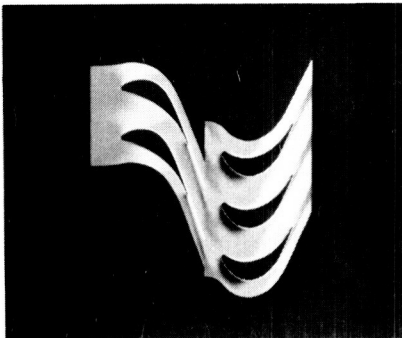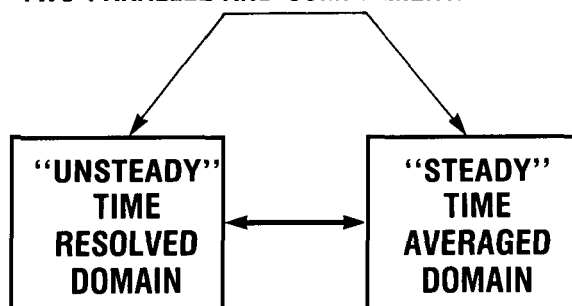
CD-87-28977

The theme of this presentation is that everything in a turbomachine is fundamentally unsteady (i.e., time dependent). This time dependency is further complicated by strong random disturbances. We have a choice. We can work in the time domain, which is expensive and time consuming; or we can work in the time-averaged domain, which is cheaper but yields less information. Furthermore, it is not simply an on/off, unsteady/steady situation. The averaging is by steps. Some of the time dependent nature can be averaged out, while some can remain. It is a matter of the time scale (or for that matter the length scale) over which the averaging is performed. Time average does not necessarily mean time independent or "steady". Thus, the question becomes both when and when not to average, and also how to average. A proper balance is required. Ultimately the engineer/designer is most interested in average information, but to get there one must properly handle time.

THEME

EVERYTHING IN A TURBOMACHINE IS UNSTEADY
(i.e., TIME DEPENDENT)

THE APPROACH TO RESEARCH IS ALONG
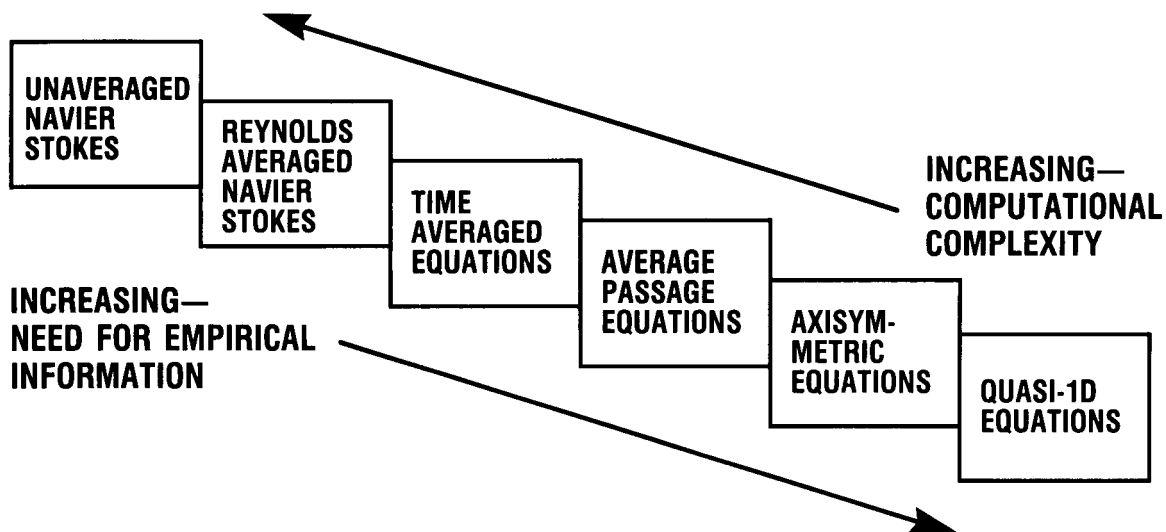TWO PARALLEL AND COMPLEMENTARY PATHS

"UNSTEADY"
TIME
RESOLVED
DOMAIN

"STEADY"
TIME
AVERAGED
DOMAIN

THE QUESTION FOR RESEARCH IS BALANCE

WHEN TO WORK IN THE UNAVERAGED TIME
DOMAIN—AND WHEN AND HOW TO TIME AVERAGE?

CD-87-28978

The time-accurate, unsteady Navier-Stokes and energy equations, which are capable of resolving all relevant time scales, describe the flow in a turbomachine. These are at the top of the stairs. They are the easiest set to formulate, but the most difficult set to solve because they require enormous computer power for the simplest cases. To ease the solution a variety of averages are taken. The critical step is to average properly by using the proper time and length scales. Each averaging step results in a loss of information or resolution in the equations, introduces more unknowns, and requires external input (obtained by experiment and by solving the exact equations for simpler cases). One can use engineering judgement to determine how much information is needed to complete the mathematical description of the particular problem at hand. The averaged equations allow this introduction of engineering judgement. The pure unaveraged Navier-Stokes equations must be solved in their entirety.

LEVELS OF COMPLEXITY ALONG THE PATH
OF COMPUTATIONAL ANALYSIS

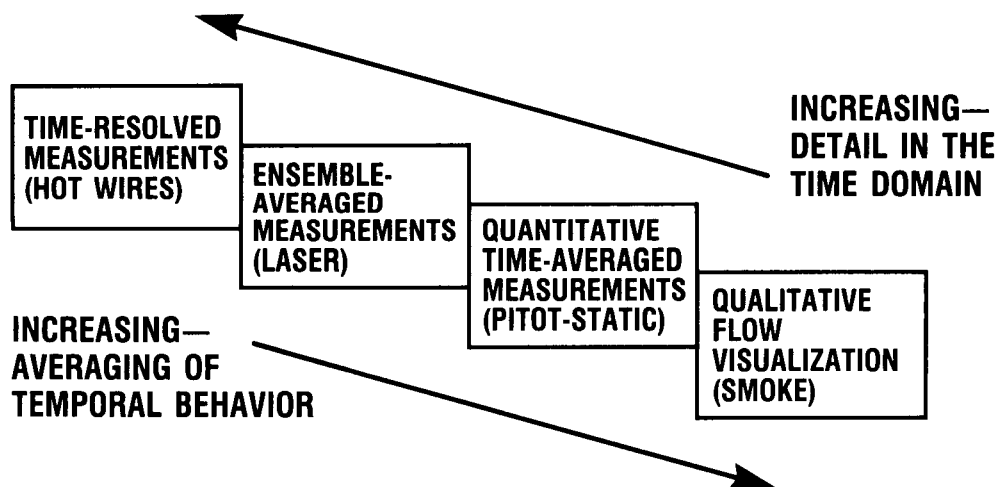UNAVERAGED NAVIER STOKES

REYNOLDS AVERAGED NAVIER STOKES

TIME AVERAGED EQUATIONS

AVERAGE PASSAGE EQUATIONS

AXISYM-METRIC EQUATIONS

QUASI-1D EQUATIONS

INCREASING—COMPUTATIONAL COMPLEXITY

INCREASING—NEED FOR EMPIRICAL INFORMATION

CD-87-28979

Both time-resolved and time-averaged measurements are essential to a full understanding of turbomachinery flow and heat-transfer characteristics. As with analysis, the problem is in determining which technique to apply and when to apply it. Frequently, an average result offers more insight than time-accurate detail. A qualitative visual observation may be crucial to understanding the essential physics. At other times, such a measurement may bury the essential physics. Traditionally, dynamic measurements have provided less accurate absolute measurements than average measurements. Recent improvements suggest that averaging time-resolved measurements may be more accurate than making average measurements, especially in heat transfer. Laser anemometry, in addition to offering the well-known nonintrusive advantage, has an especially nice feature of providing ensemble averages of the random statistics while retaining and identifying the deterministic time dependency. Ultimately, all levels are needed to do the job. The challenge is to choose the right tool for the right job.

LEVELS OF COMPLEXITY ALONG THE PATH OF EXPERIMENTAL MEASUREMENTS

CD-87-28980

3-33

The average passage equation system, developed by J. Adamczyk of NASA Lewis, is an example of averaging the Navier-Stokes equations. The three momentum equations and the energy equation are subjected to three averaging steps: one to remove random unsteadiness, a second to remove unsteadiness associated with blade-passing frequency and, finally, one to account for the uneven airfoil count from row to row. At its most fundamental level the averaging process introduces 11 unknowns in the axial momentum equation alone. The advantage is that the resulting equation set is much easier to solve mathematically. It will be necessary to conduct physical and/or numerical experiments to provide the correlations which bring closure to these equations. The properly averaged equations provide the framework for a large research effort into understanding the physics of fluid flow and heat transfer in turbomachines.

AVERAGE PASSAGE FRAMEWORK OF EQUATIONS

AXIAL MOMENTUM EQUATION:

$$\frac{\partial}{\partial t_1}\lambda_i\,\bar{\bar{\rho}}\,r\tilde{V}_z + \frac{\partial}{\partial r}\lambda_i\,r\bar{\bar{\rho}}\tilde{V}_r\tilde{V}_z + \frac{\partial}{\partial\theta}\lambda_i\,\bar{\bar{\rho}}\tilde{V}_\theta\tilde{V}_z + \frac{\partial}{\partial z}\lambda_i\left(r\bar{\bar{\rho}}\tilde{V}_z\tilde{V}_z + r\bar{\bar{P}}\right)$$

CONVECTIVE TERMS

TRANSIENT TERM

$$= \frac{\partial}{\partial r}\lambda_i\left(r\bar{\bar{\tau}}_{rz} - r\,\bar{\bar{\rho}}\,\hat{\tilde{V}}_r\hat{\tilde{V}}_z - r\,\bar{\bar{\rho}}\,\widetilde{V}_r\widetilde{V}_z - r\rho V_r'V_z'\right)$$

$$+ \frac{\partial}{\partial\theta}\lambda_i\left(\bar{\bar{\tau}}_{\theta z} - \bar{\bar{\rho}}\,\hat{\tilde{V}}_\theta\hat{\tilde{V}}_z - \bar{\bar{\rho}}\,\widetilde{V}_\theta\widetilde{V}_z - \rho V_\theta'V_z'\right)$$

DIFFUSIVE TERMS

WHERE

BODY FORCE TERMS

$$+ \frac{\partial}{\partial z}\lambda_i\left(r\bar{\bar{\tau}}_{zz} - r\,\bar{\bar{\rho}}\,\hat{\tilde{V}}_z\hat{\tilde{V}}_z - r\,\bar{\bar{\rho}}\,\widetilde{V}_z\widetilde{V}_z - r\rho V_z'V_z'\right)$$

$$+ F_{IN}^{(ZR)} + F_V^{(ZR)} + F_{IN}^{(ZS)} + F_V^{(ZS)}$$

THE SUCCESSIVE OVERBARS
REPRESENT AVERAGING OUT:
(1) RANDOM UNSTEADINESS
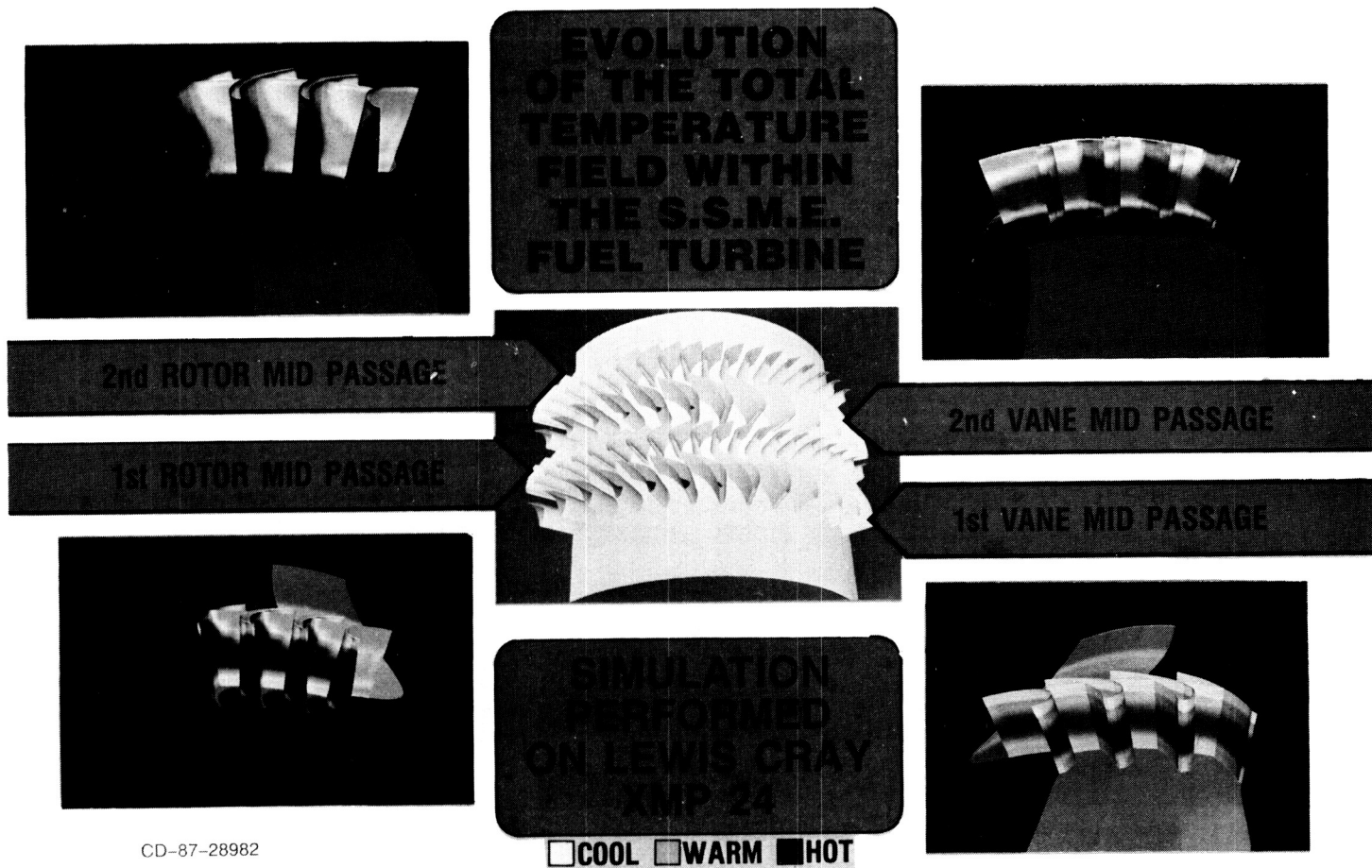(2) PERIODIC UNSTEADINESS
(3) UNEQUAL BLADE COUNT

> **THE AVERAGING YIELDS A NEW EQUATION CONTAINING 11 UNKNOWNS WHICH HAVE TO BE DETERMINED BY CONDUCTING PHYSICAL AND/OR NUMERICAL EXPERIMENTS**

CD-87-28981

EVOLUTION OF THE TOTAL-TEMPERATURE FIELD WITHIN THE SPACE SHUTTLE MAIN
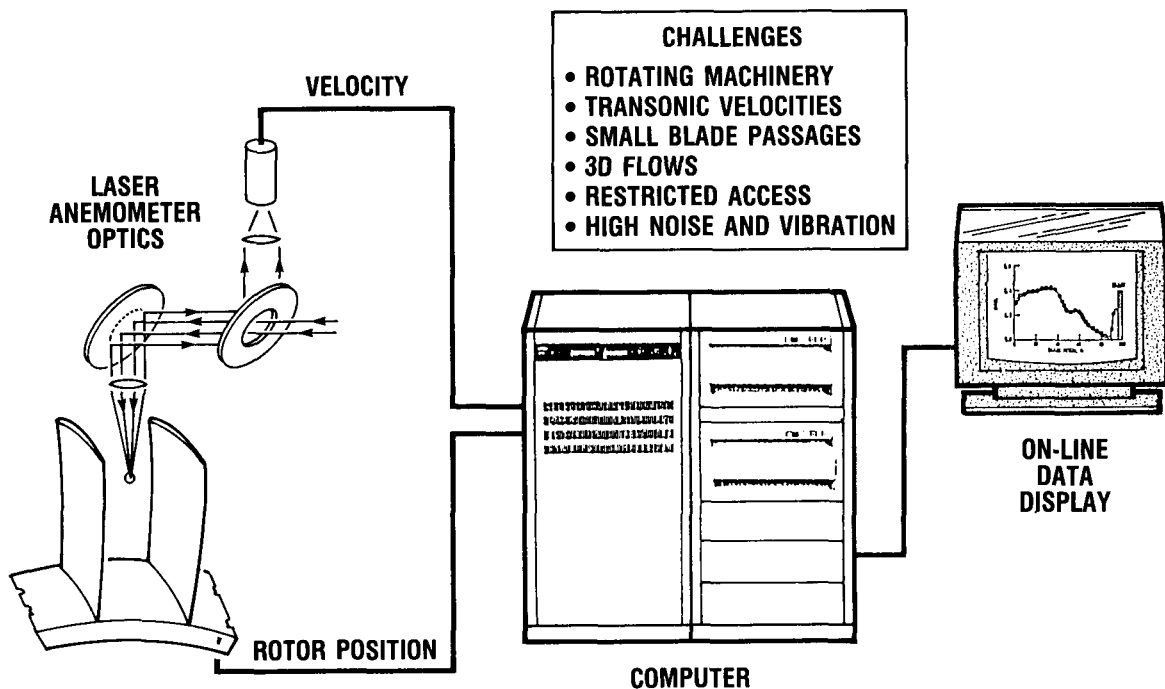ENGINE (SSME) FUEL TURBINE

An example of a calculation using the average passage system of equations depicts
the evolution of a total-temperature distortion through the fuel pump turbine. The
total-temperature profiles are color coded with red denoting regions of hot gas and
blue denoting regions of cold gas. At this stage in the development of the average
passage equation system many of the unknown viscous correlations remain to be
determined and are set equal to zero for this calculation. The equations do,
however, properly account for the interaction between blade rows. These results
show that inviscid mixing as a result of streamwise vorticity generation can produce
significant temperature differences between the suction and pressure side of a
blade. This difference can lead to local regions of high thermal stress which can
cause blade failure. The ability to capture the physics associated with this
inviscid mixing process is a key element in increasing the durability of turbine
blading.

EVOLUTION OF THE TOTAL TEMPERATURE FIELD WITHIN THE S.S.M.E. FUEL TURBINE

2nd ROTOR MID PASSAGE

1st ROTOR MID PASSAGE

2nd VANE MID PASSAGE

1st VANE MID PASSAGE

SIMULATION PERFORMED ON LEWIS CRAY XMP-24

☐COOL ☐WARM ■HOT

CD-87-28982

High-speed turbomachinery research facilities are characterized by high-speed rotating machines, small blade-passage heights, three-dimensional flows, transonic velocity levels, high noise and vibration levels, and restricted mechanical access. Because of its high spatial and temporal resolution and nonintrusive nature, laser anemometry has become the measurement method of choice for obtaining the detailed data required to assess the accuracy and sensitivity of flow analysis codes. In order to overcome the long measurement times required by laser anemometers and to capitalize on the detailed nature of the data which they generate, computer control of data acquisition and real-time data reduction and display are required. NASA Lewis is recognized as a world leader in the development and application of computer-controlled laser anemometer systems for use in both rotating and nonrotating turbine and compressor research applications.

TURBOMACHINERY LASER ANEMOMETRY SYSTEMS

VELOCITY

LASER
ANEMOMETER
OPTICS

CHALLENGES
- **ROTATING MACHINERY**
- **TRANSONIC VELOCITIES**
- **SMALL BLADE PASSAGES**
- **3D FLOWS**
- **RESTRICTED ACCESS**
- **HIGH NOISE AND VIBRATION**

ON-LINE
DATA
DISPLAY

ROTOR POSITION

COMPUTER

CD-87-28983

FAN ROTOR VORTEX SHEDDING – DEVELOPING MODELS TO EXPLAIN FLOW PHYSICS

Karman vortex streets are known to exist in blunt-body wakes over a wide flow
regime. However, the existence of vortex streets in transonic fan and compressor-
blade wakes was not generally anticipated since these blades have thin trailing
edges. Laser anemometer measurements obtained in the wake of a transonic fan blade
indicated two distinct states of the flow in the central portion of the blade wake –
a high-velocity state and a low-velocity state. This behavior is consistent with
that which would be displayed by a Karman vortex street. A simple vortex street
model was constructed in an attempt to explain the experimental measurements. The
model qualitatively agreed with the bimodal character of the velocity measurements.
The model was also used to explain, for the first time, the highly unsteady nature
of high-response pressure measurements made in the same wake flowfield. This
research, which was a cooperative effort with MIT, typifies the manner in which
advanced measurements coupled with simple modelling improve our understanding of
complex flow phenomena.

FAN ROTOR VORTEX SHEDDING
DEVELOPING MODELS TO EXPLAIN FLOW PHYSICS

**LASER MEASUREMENT HISTOGRAMS
SUGGEST PRESENCE OF VORTEX SHEDDING**
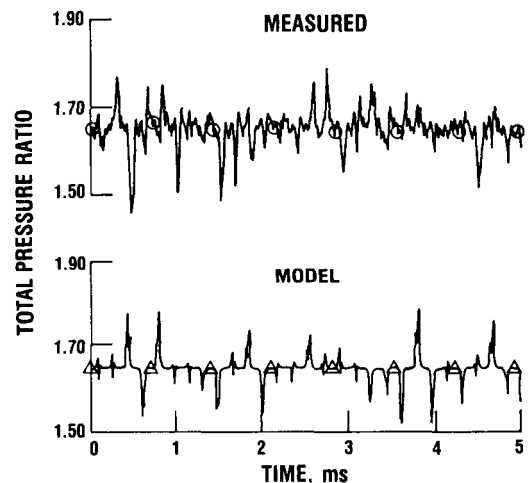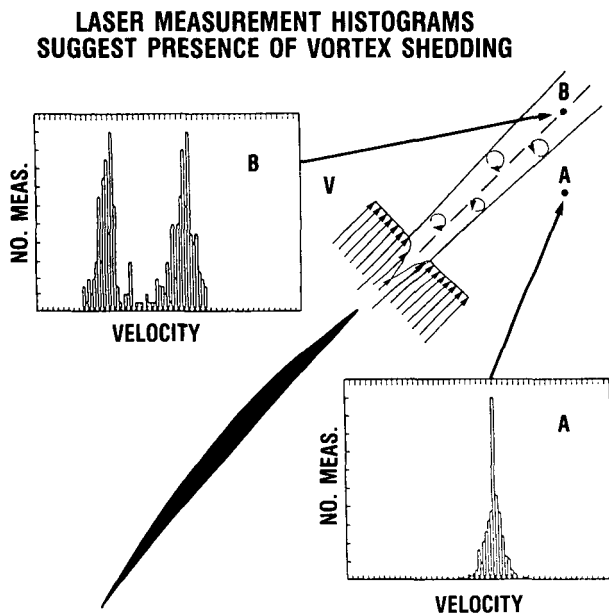
VORTEX WAKE MODEL DEDUCED FROM
THE LASER MEASUREMENTS AND
COMPARED TO HIGH-RESPONSE
PRESSURE PROBE DATA

CD-87-28984

3-37

Measurements of the unsteady flowfield within a compressor stator operating
downstream of a transonic fan rotor have been obtained using laser anemometry. The
figure shows a contour plot of the ensemble-averaged unresolved unsteadiness in the
stator (which includes unsteadiness due to both turbulence and vortex shedding) for
one rotor/stator relative position. Areas of high unresolved unsteadiness contain
fluid which is in the rotor-blade wake. As the rotor blades rotate past the stator
blades, the rotor wakes are convected through the stator row by the absolute flow
velocity and, subsequently, chopped by the stator blades. Data obtained at
additional times during the blade-passing cycle have been used to produce a movie
sequence which illustrates the ensemble-averaged wake dynamics and its effect on the
stator flowfield. We have the tools to measure temporal behavior in generic
equivalents of a real machine.

COMPRESSOR "TURBULENT" KINETIC ENERGY DISTRIBUTION

ONE FRAME FROM DATA MOVIE SHOWING THE PROGRESS OF A WAKE
THROUGH A STATOR PASSAGE DOWNSTREAM OF THE ROTOR

CD-87-28986

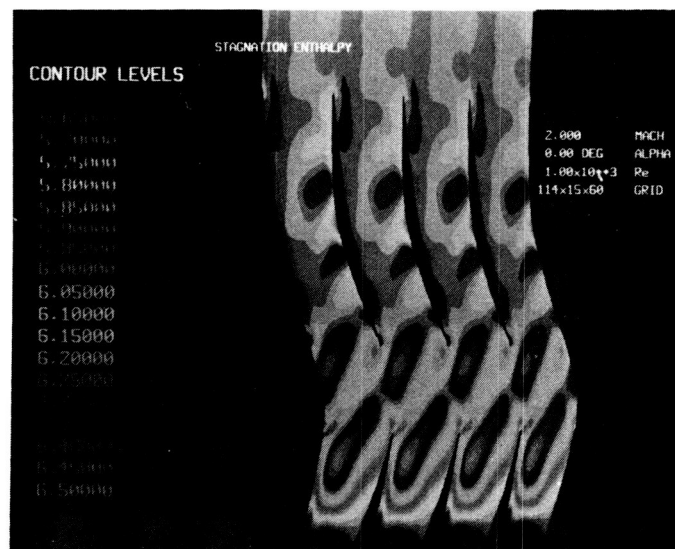UNSTEADY EULER CALCULATION IN A SUPERSONIC THROUGHFLOW FAN

A simulation was performed by David Whitfield and Mark Janius of Mississippi State University, in collaboration with NASA Lewis researchers, of the flow through a supersonic throughflow fan stage. This machine differs from today's machinery in that the axial Mach number is supersonic. For supersonic cruise, it offers an improvement in performance over transonic machinery. The three-dimensional simulation highlighted the rotor-stator interaction which occurs at design operating conditions. One of the interactions under study is the formation of "hot spots" at the leading edge of the stator. An animation illustrating the processes will be shown later in this conference. The "hot spots" are at a temperature which exceeds the instantaneous total temperature (absolute) of the flow stream exiting the rotor. Their formation is believed to be caused by the motion of the shock wave eminating from the pressure surface of the stator. We have the tools to compute temporal behavior in a real machine.

UNSTEADY EULER CALCULATION FOR A SUPERSONIC THROUGHFLOW FAN

ONE FRAME FROM A COMPUTATIONAL MOVIE SHOWING THE PROGRESS OF THE SUPERSONIC ROTOR EXIT TOTAL TEMPERATURE THROUGH THE DOWNSTREAM STATOR
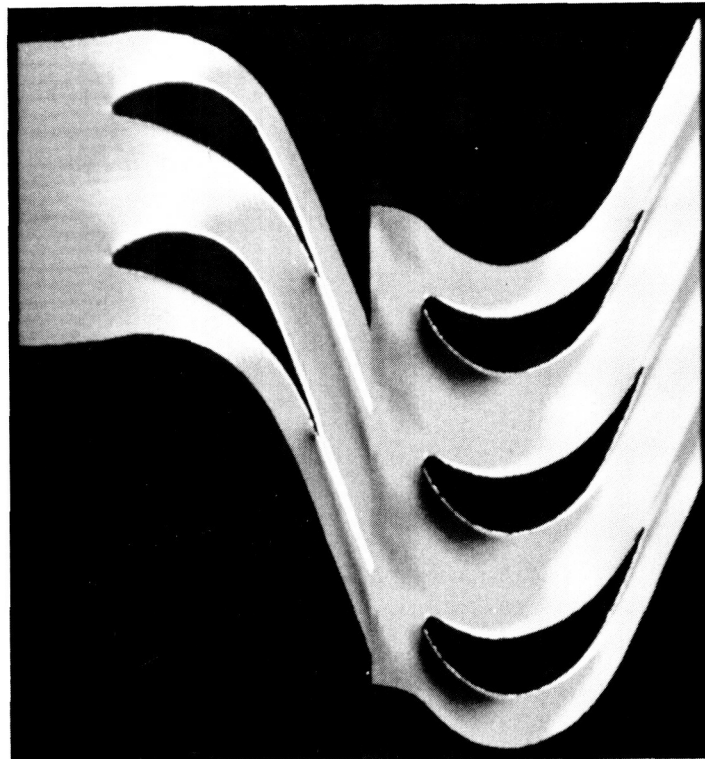

CD-87-28987

A NASA Lewis quasi-three-dimensional viscous code used to solve for the flow in an isolated turbomachinery blade row was modified to handle equal pitch stator/rotor interaction computations. The solution procedure has been applied to the first-stage turbine rotor of the space shuttle main engine (SSME) fuel turbopump. For this calculation, the upstream stator was scaled so that its pitch matched that of the rotor, and the pitch-to-chord ratio remained unchanged. A converged periodic solution was obtained after the stator had seen 10 passing rotor blades or 10 pitch rotations of the rotor, which takes about 2.5 hours on the Cray. Mach contours are shown in the figure for an equal pitch stator/rotor configuration. The average inlet Mach number to the upstream stator is 0.15. The wake region that develops behind the stator passes through the grid interface and is seen in the rotor computational domain. Currently, the analysis is being applied to multiple passages of a single-stage turbine. The stage airfoil configuration is two upstream stators followed by three rotors. The airfoil geometries are taken from the first stage of the SSME fuel turbopump and are scaled to 2:3 from their actual 41:63 airfoil ratio. The details of such interactions are important to our understanding of turbomachinery flows.

ROTOR/STATOR INTERACTION CALCULATION—SSME FUEL TURBINE

TIME ACCURATE 2D NAVIER-STOKES CALCULATION, SHOWING THE MACH CONTOUR DISTRIBUTION AT ONE INSTANT OF TIME IN A BLADE-PASSING PERIOD

CD-87-28988

EFFECT OF WAKES ON LAMINAR-TURBULENT TRANSITION IN A TURBINE STAGE

Detailed phase-resolved heat-flux data have been obtained on the blade of a full-stage rotating turbine (Teledyne 702) in work being done at CalSpan Corp. by Michael Dunn for Teledyne Corp. in conjuntion with an Air Force-sponsored Vane-Blade Interaction program. A shock tube is used as a short-duration source of heated air, and platinum thin-film gages are used to obtain the heat-flux measurements. Some thin-film gages can be seen in the right-hand side of this figure in a leading-edge insert. Heat-flux results are presented in the left-hand side of the figure for the midspan at several locations along the suction surface from stagnation point to 78-percent wetted distance. Each phase-resolved plot represents the ensemble-average of about four to five vane-wake (or passage) crossings. The rapid decrease in heat flux level from stagnation point to trailing edge is evident, as well as the fluctuating laminar-to-turbulent (and back to laminar) component, as the rotor cuts the stator wakes. This high-frequency oscillation from laminar-to-turbulent flow has important implications for turbine heat-transfer analysis and can only be identified with sophisticated sensors, electronics, and dynamic signal analysis.

EFFECT OF WAKES ON LAMINAR-TURBULENT TRANSITION IN A TURBINE STAGE
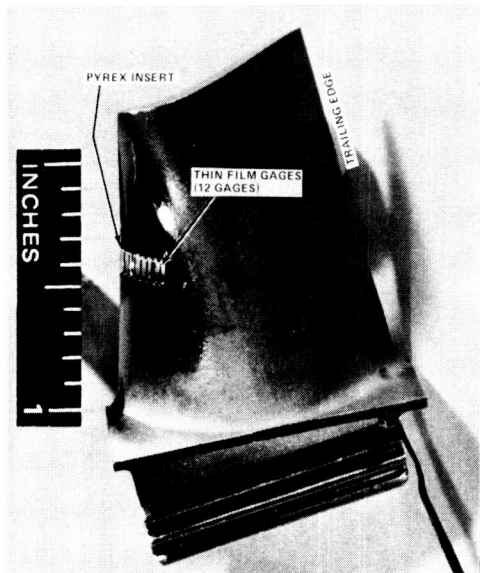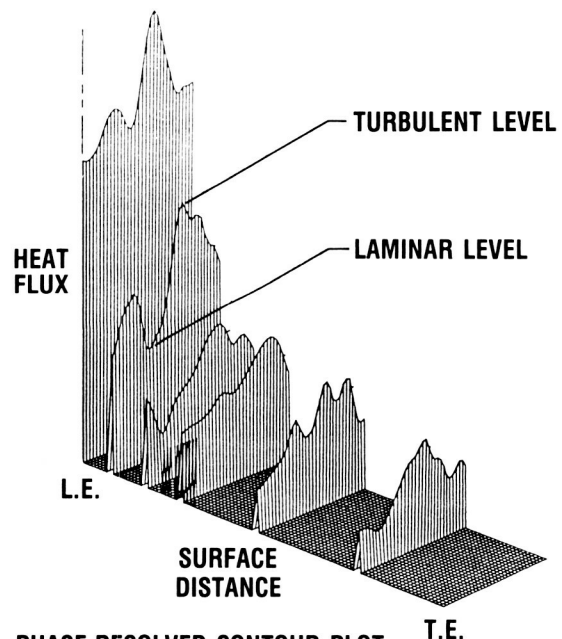

PHOTO OF A ROTOR BLADE INSTRUMENTED
WITH THIN-FILM SENSORS

CD-87-28989

PHASE-RESOLVED CONTOUR PLOT
SHOWING THE PROGRESSION OF
TRANSITION ALONG THE AIRFOIL

As a result of several recent studies, the nature and significance of wake-related flow unsteadiness in turbomachinery blading and its profound effect on heat transfer are beginning to be recognized. In particular, the enhancement of heat transfer to the stagnation region is of interest because of the critical importance of heat transfer in this region. The effect of freestream turbulence on time-average stagnation region heat transfer has been well documented. However, very few measurements have been obtained of the time-resolved effects of wake passage on heat transfer and the relationship of these effects to the corresponding velocity fluctuations. Current efforts are aimed at obtaining such measurements and performing statistical analysis to determine correlations between the unsteady velocity and heat-flux records. The figure shows a schematic of the rotor-wake rig and representative fluctuating velocity measurements. In addition, stagnation region unsteady heat-flux records are shown which reveal the high degree of heat transfer enhancement associated with each wake-passing event.
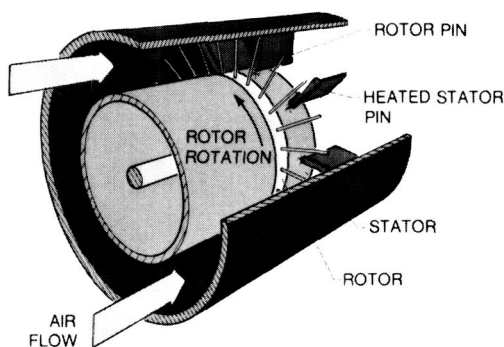
UNSTEADY HEAT TRANSFER IN ROTOR-WAKE FLOWS

SIMULTANEOUS HEAT FLUX RECORDS

INSTANTANEOUS VELOCITY RECORD
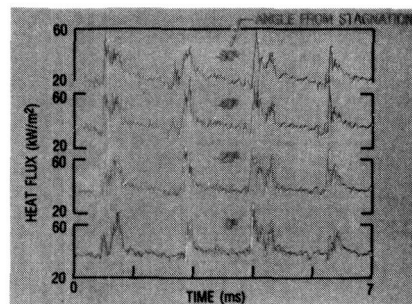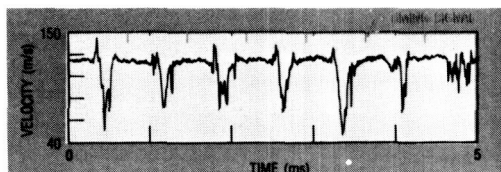
ROTATING SPOKED WHEEL PRODUCES GOOD SIMULATION OF AN AIRFOIL TRAILING-EDGE WAKE

ROTOR PIN

HEATED STATOR PIN

ROTOR ROTATION

STATOR

ROTOR

AIR FLOW

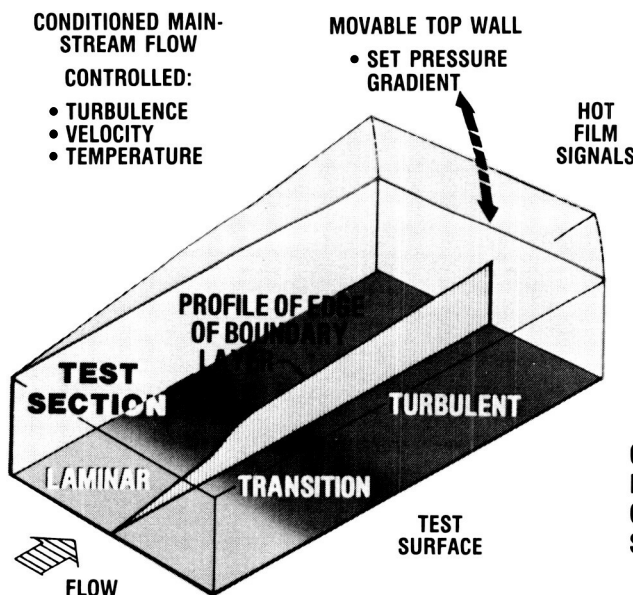**INSTANTANEOUS HEAT FLUX IN STAGNATION REGION IN WAKE OF SPOKED ROTOR**

CD-87-28990

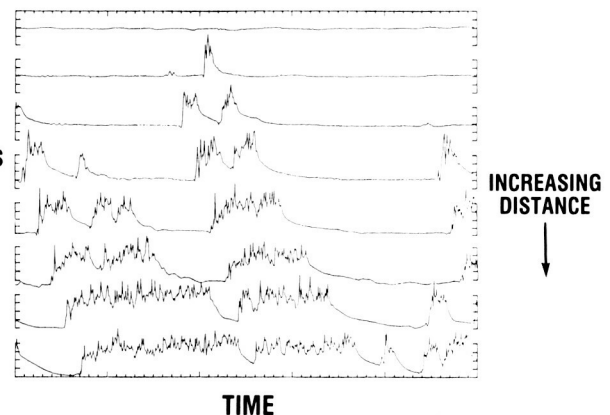BOUNDARY-LAYER TRANSITION RESEARCH -- A STUDY OF INTERMITTENT BEHAVIOR

Since gas-side heat-transfer coefficients can vary by an order of magnitude within the transition region, a detailed understanding of boundary-layer transition is critical to the design of effective turbine-airfoil cooling schemes. This is particularly important when one realizes that such events can occur at blade-passing frequency. The present research program is aimed at understanding the fundamental differences between the "classical" boundary-layer transition process and the "bypass" transition process which occurs when a laminar boundary layer is perturbed by large freestream disturbances. The time-resolved hot wire velocity measurements obtained at a grid-generated freestream turbulence level of only 0.7 percent, indicate a bypass of small disturbances and the rapid development of a turbulent boundary layer as the flow progresses down the plate.

BOUNDARY-LAYER TRANSITION RESEARCH
A STUDY OF INTERMITTENT BEHAVIOR

SKETCH OF TEST SECTION SHOWING MAJOR FEATURES OF FACILITY

FREE-STREAM TURBULENCE LEVEL 0.77%

CONDITIONED MAIN-STREAM FLOW

MOVABLE TOP WALL
• SET PRESSURE GRADIENT

CONTROLLED:
• TURBULENCE
• VELOCITY
• TEMPERATURE

HOT FILM SIGNALS

INCREASING DISTANCE

PROFILE OF EDGE OF BOUNDARY LAYER

TEST SECTION

TURBULENT

LAMINAR   TRANSITION

TEST SURFACE

FLOW

TIME

CARPET PLOT OF SIMULTANEOUS HOT FILM RECORDS, SHOWING PROGRESSION AND GROWTH OF TURBULENCE ALONG THE PLATE SURFACE

CD-87-28991

The photograph of the experimental color temperature patterns shows contours of constant temperature obtained from an experimental technique in which a uniformly heated turbine-vane cascade endwall surface is operated in an air flow. Resulting isotherms on the test surface (which are also lines of constant heat-transfer coefficient) are indicated by thermochromic liquid crystals. The photographic data was then digitized for computer-based processing and display, to show color contours of Stanton Number (nondimensional heat-transfer coefficient). The highest heat transfer rates occur in the vane stagnation region (shown in red). This computer-generated display can be used for direct comparison with code-generated predictions. Complex phenomena, such as these, require complex analyses — the three-dimensional Navier-Stokes codes.

HIGH-RESOLUTION LIQUID-CRYSTAL TURBINE ENDWALL HEAT TRANSFER DATA

COMPUTER-BASED COLOR CONTOURS OF THE ENDWALL HEAT TRANSFER DISTRIBUTION

EXPERIMENTAL LIQUID-CRYSTAL COLOR TEMPERATURE PATTERNS

(ISOTHERMS AT CONSTANT HEAT FLUX)
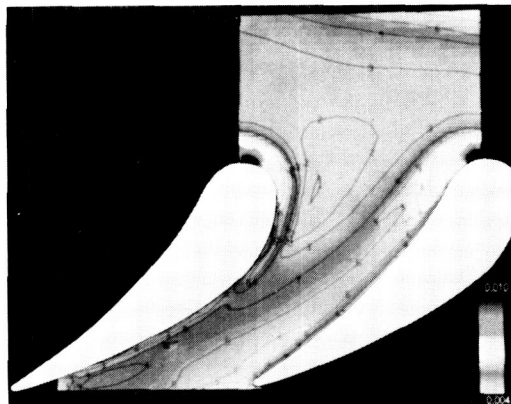
(DIGITIZED COMPOSITE OF EXPERIMENTAL DATA)

CD-87-28993

A three-dimensional Navier-Stokes analysis code (RVC3D) is being developed for turbomachinery blade rows. The Navier-Stokes equations are written in a body-fitted coordinate system rotating about the x-axis. Streamwise viscous terms are neglected by using the thin-layer assumption, and turbulence effects are modeled with the Baldwin-Lomax eddy viscosity. The equations are discretized by using finite differences, and solved by using a multistage Runge-Kutta algorithm with a spatially varying time step and implicit residual smoothing. Calculations have been made of a horseshoe vortex formed at the junction of a turbulent endwall boundary layer and a blunt fin. This geometry is to be tested experimentally later this year by members of the Heat Transfer Branch. The calculations were done on a 65 by 33 by 25 grid (lower figure) at the nominal tunnel operating conditions (Mach number = 0.6, fin thickness Reynolds number = 260 000, inlet boundary-layer thickness = 1 in). Total pressure contours, 0.025 in. above the endwall, show the primary vortex core (center figure). Vector plots on the upstream symmetry plane (upper left) and on a downstream cross-channel plane (upper right) show the development of a double vortex system. The calculations required about one million words of storage and 10 min of CPU time on a Cray X-MP.

3D NAVIER-STOKES CODE ANALYSIS OF THE
TURBULENT HORSESHOE VORTEX

CALCULATION OF HORSESHOE VORTEX FOR SAME GEOMETRY AND CONDITIONS OF THE COMPRESSIBLE FLOW TUNNEL

FLOW VECTORS

STAGNATION POINT

DOWNSTREAM

TOTAL PRESSURE CONTOURS

GRID
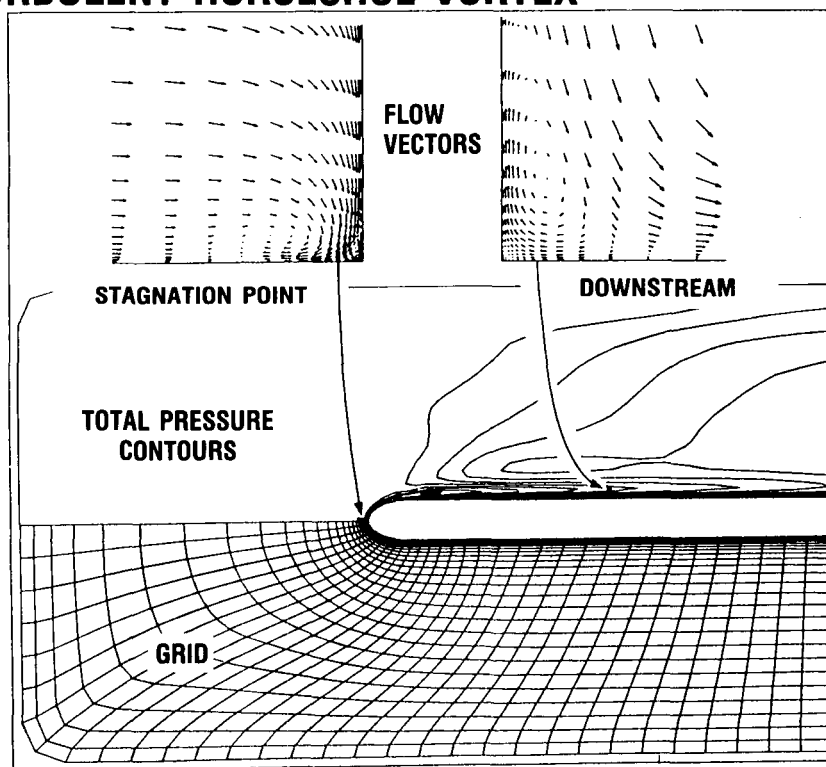
CD-87-28994

This new facility for fluid mechanics and heat-transfer research will provide benchmark quality experimental data for internal flow code validation. Focus will be on the three-dimensional interaction of the intersecting model with the surface plate (i.e., the horseshoe vortex). The facility will have the following capabilities: (1) maximum Mach number of 0.6; (2) no tunnel side-wall boundary layers; (3) controllable boundary layer on top and bottom walls of the tunnel; and (4) low inlet turbulence (less than 0.5 percent). The first phase of the experiment will consist of three parts: (1) fluid mechanics measurements will be taken by using a five-hole probe and hot film shear-stress gauges, (2) various flow visualization techniques will be used to define the flow path at the intersecting surfaces, and (3) heat transfer data will be recorded by the liquid crystal technique. The second phase of the work, to be conducted in about two years, will include full flowfield measurements by laser anemometry. This experiment and the RVC3D code development form a critical partnership to the successful development of a turbomachinery analysis capability.

3D COMPRESSIBLE FLOW TUNNEL
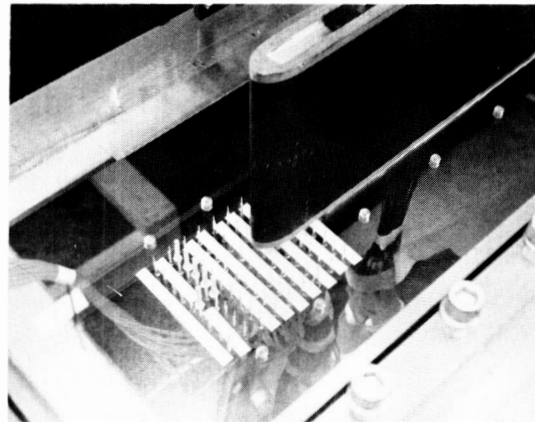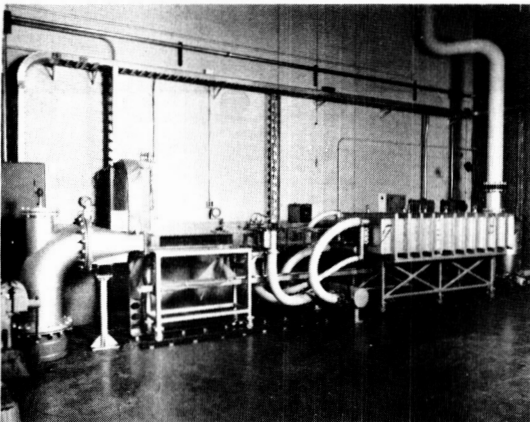
OBJECTIVE

TO OBTAIN DATA TO VERIFY COMPUTATIONAL FLUID MECHANICS COMPUTER CODES THAT ARE CAPABLE OF SOLVING FULLY 3D FLOWS INCLUDING HEAT TRANSFER

CD-87-28995

LARGE LOW-SPEED CENTRIFUGAL COMPRESSOR, NEW FLOW-PHYSICS AND CODE-VALIDATION RIG

Centrifugal compressors feature large surface area and small exit-passage heights. Viscous flow effects, therefore, have a major impact on the flowfield within centrifugal compressors. The inability to accurately predict and measure these flows contributes in large part to the inherently lower efficiency of centrifugal compressors relative to axial-flow compressors.

The large low-speed centrifugal compressor shown in the background of the photograph has been designed specifically to provide flow modelling and viscous code-validation data for centrifugal compressors. The impeller was designed to be aerodynamically similar to high-performance, high-speed centrifugal compressors such as the small 6:1 pressure-ratio impeller shown in the photograph. The low-speed impeller has a tip diameter of 50 inches and a rotational speed of 1950 rpm. Inlet and exit blade heights are 9 inches and 4.75 inches, respectively. The large size and low speed of the new impeller generate viscous flow regions (such as blade and endwall boundary layers) and tip clearance flows which are large enough to measure in detail with laser anemometry.

LARGE LOW-SPEED CENTRIFUGAL COMPRESSOR
NEW FLOW PHYSICS AND CODE VALIDATION RIG

UNDERSTANDING THE FLOW PHYSICS AND THE VALIDATION OF COMPLEX TURBOMACHINERY 3D NAVIER-STOKES CODES REQUIRES VERY LARGE ROTATING MACHINERY IN ORDER TO BE ABLE TO EXAMINE THE VISCOUS BOUNDARY LAYERS

SIZE COMPARISON BETWEEN LOW-SPEED IMPELLER AND 2-LB/S HIGH-SPEED IMPELLER

CD-87-28996

An efficient Navier-Stokes analysis code (RVCQ3D) has been developed for turbomachinery. The effects of radius change, stream surface thickness, and rotation are included, which allows calculations of centrifugal impellers, radial diffusers, and axial machines with contoured endwalls. The unsteady Navier-Stokes equations are solved in finite-difference form using an explicit Runge-Kutta algorithm with a spatially varying time step and multigrid convergence acceleration. The flow in a 6:1 pressure-ratio centrifugal impeller has been calculated on a 161 by 33 grid. Relative Mach number contours in the figure show a supersonic bubble on the leading edge terminated by a shock. Rotational effects make the suction-surface boundary layer thin, the pressure-surface boundary layer thick, and they cause the wake to leave the trailing edge in a spiral. The calculations required about 3 000 000 words of storage and 1.5 min of CPU time on a Cray X-MP. The flow in the new, large low-speed rig has also been calculated and is shown on the right. The ability to validate this and other codes in the large low-speed rig will generate confidence in the high-speed calculations where validation is near to impossible.

QUASI-3D NAVIER-STOKES CODE FOR TURBOMACHINERY ANALYSIS
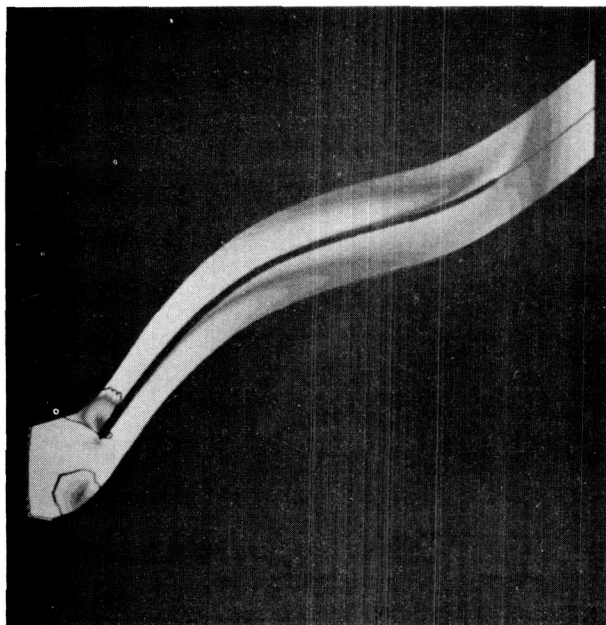
ANALYSIS OF HIGH-SPEED 6:1
CENTRIFUGAL IMPELLER

BLADE THICKNESS ≈ 0.5 IN
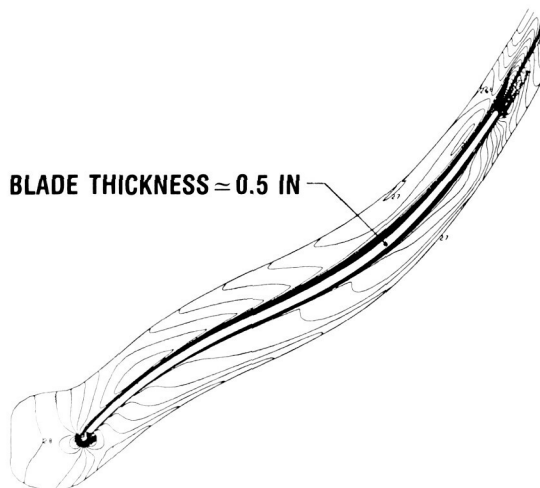
ANALYSIS OF LARGE LOW-SPEED
CENTRIFUGAL IMPELLER

CD-87-28997

In order to fully validate the codes, they must be shown capable of capturing real physics in real machines. An example is the shock behavior in a transonic fan. Many axial fan and compressor design systems currently in use do not account for passage shock three-dimensionality. In addition, preliminary blade designs are often performed as two-dimensional calculations on blade-to-blade stream surfaces. An assessement of the shock three-dimensionality in transonic rotors is therefore necessary in order to properly account for three-dimensional effects in the design process. Shock locations determined from laser anemometer measurements are shown on blade-to-blade surfaces of revolution in the upper half of the figure. Three different views of the same data, as displayed on a graphics workstation, are shown in the lower half of the figure. A significant spanwise lean of the shock surface is evident in these three-dimensional views.

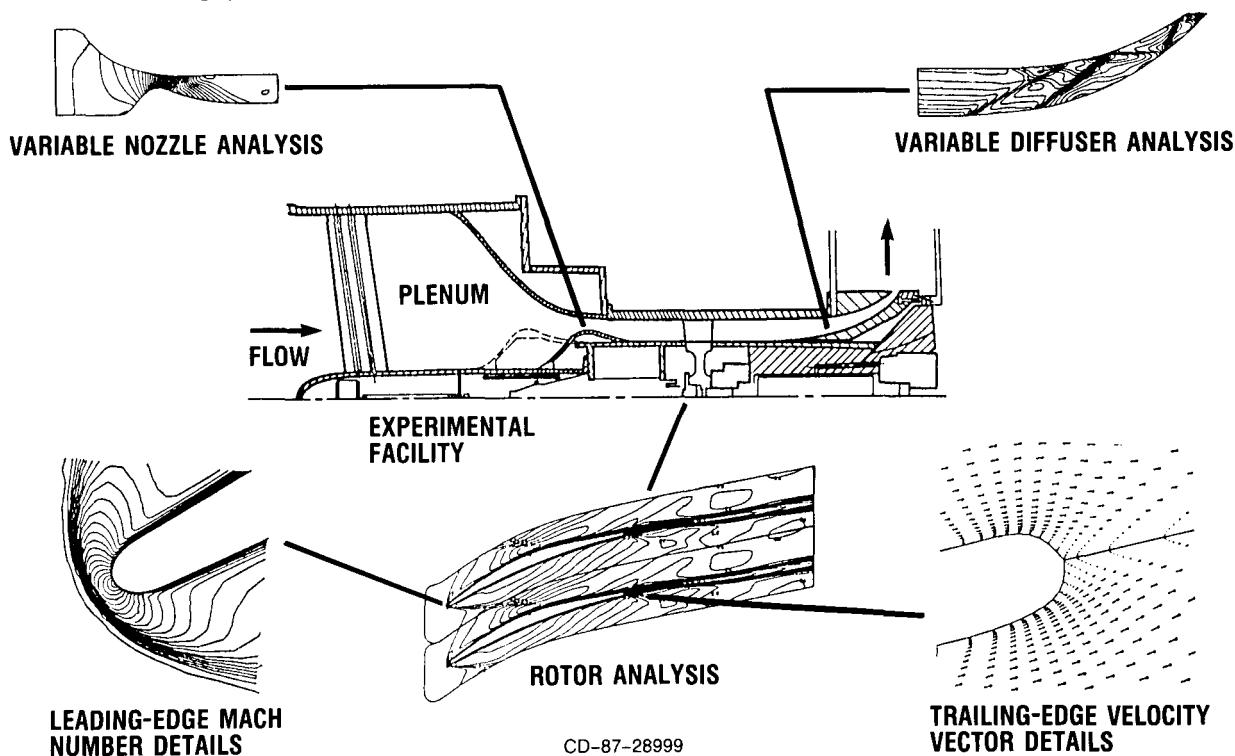## MEASUREMENT OF 3D SHOCK STRUCTURE IN A TRANSONIC AXIAL-FLOW FAN
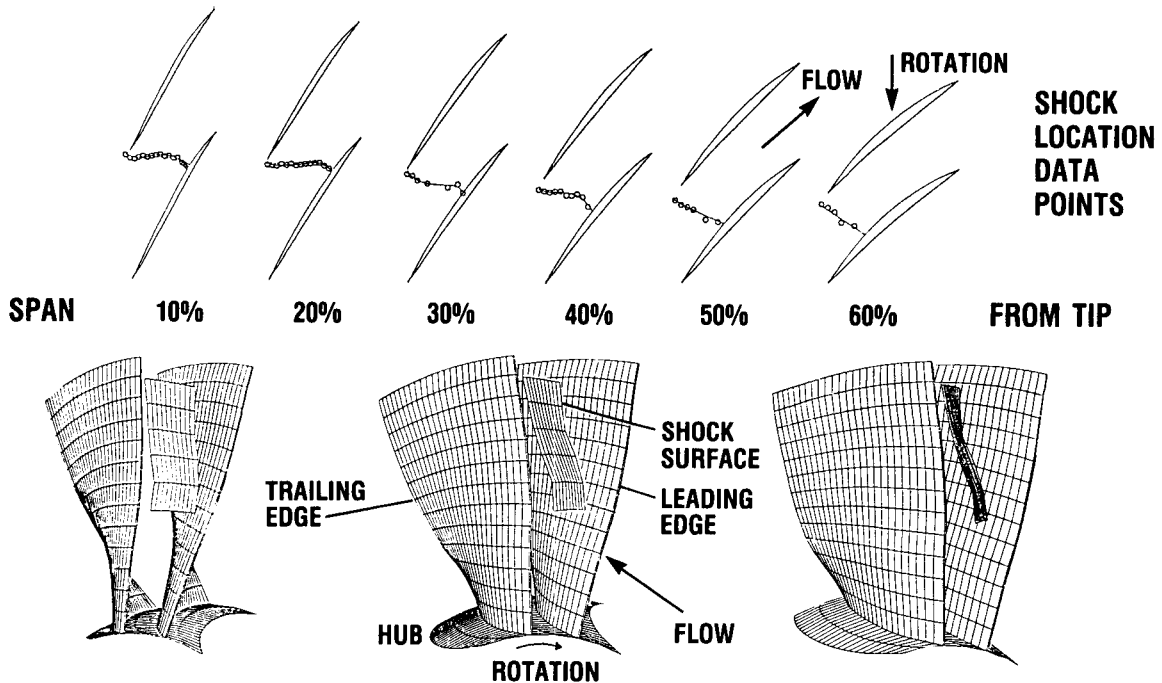
DATA ENHANCEMENT AND 3D VIEWING ROTATION

CD-87-28998

Increased emphasis on sustained supersonic cruise or hypersonic cruise has revived interest in the supersonic throughflow fan as a possible component in advanced propulsion systems. Use of a fan that can operate with a supersonic inlet axial Mach number is attractive from the standpoint of reducing the inlet losses incurred in diffusing the flow from a supersonic flight Mach number to a subsonic one at the fan face. The data base for components of this type is practically nonexistent, and design of any experiment to study feasibilty of this concept must rely heavily upon advanced computational tools to enhance the possibility of success. Computer codes that have been developed for design and analysis of transonic turbomachines were modified to allow calculations of blade rows with supersonic inlet Mach numbers. An inviscid/viscous code and a parabolized viscous code were used to design and analyze the variable nozzle and variable diffuser necessary for the experiment. Off-design analysis of the various components of the experiment indicated that all components would operate as expected over the flow and speed range of the experiment. The figure shows the results obtained for the inlet variable nozzle which sets up the inlet flowfield, the fan rotor, and the variable diffuser which decelerates the flow toward the collector inlet. All components were analyzed with two different codes in order to give increased confidence in the computed results. This is the ultimate goal of the turbomachinery program – to develop the tools which allow us to push beyond our experience with confidence.

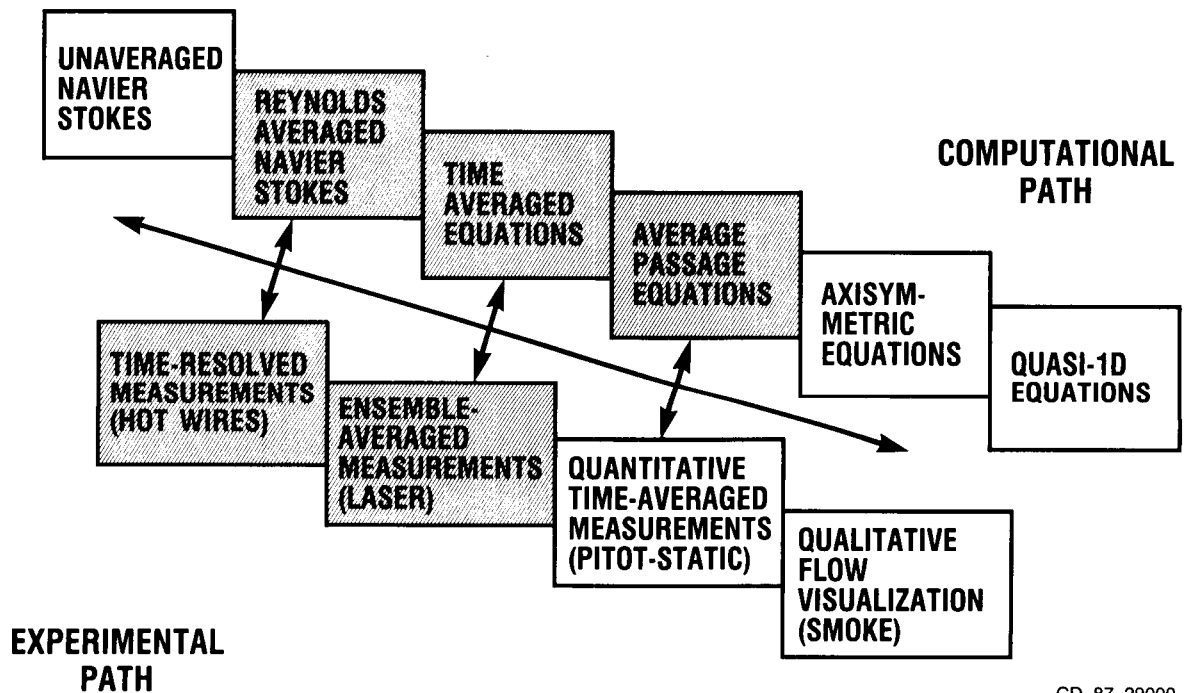## APPLICATION OF ADVANCED CODES FOR DESIGN OF SUPERSONIC THROUGHFLOW FAN EXPERIMENT

VARIABLE NOZZLE ANALYSIS

VARIABLE DIFFUSER ANALYSIS

PLENUM

FLOW

EXPERIMENTAL
FACILITY

LEADING-EDGE MACH
NUMBER DETAILS

ROTOR ANALYSIS

CD-87-28999

TRAILING-EDGE VELOCITY
VECTOR DETAILS

A complete and mature program has research at all levels along the computational and experimental paths. In the NASA LeRC turbomachinery program, special emphasis is placed on the analytic range from the ensemble (Reynolds) averaged Navier–Stokes equations to the average passage equations. The experimental emphasis is on high-response time-resolved measurements and in real machinery laser anemometry measurements. It is important to emphasize that the successful application of these tools will require a strong interaction between the computational and experimental paths.

POSITION OF THE TURBOMACHINERY PROGRAM ON COMPUTATIONAL AND EXPERIMENTAL PATHS

UNAVERAGED NAVIER STOKES

REYNOLDS AVERAGED NAVIER STOKES

TIME AVERAGED EQUATIONS

AVERAGE PASSAGE EQUATIONS

COMPUTATIONAL PATH

AXISYM-METRIC EQUATIONS

QUASI-1D EQUATIONS

TIME-RESOLVED MEASUREMENTS (HOT WIRES)

ENSEMBLE-AVERAGED MEASUREMENTS (LASER)

QUANTITATIVE TIME-AVERAGED MEASUREMENTS (PITOT-STATIC)

QUALITATIVE FLOW VISUALIZATION (SMOKE)

EXPERIMENTAL PATH

CD-87-29000

CHEMICAL REACTING FLOWS

Edward J. Mularz
and
Peter M. Sockol

ABSTRACT

Future aerospace propulsion concepts involve the combustion of liquid or gaseous fuels in a highly turbulent internal air stream. Accurate predictive computer codes which can simulate the fluid mechanics, chemistry, and turbulence-combustion interaction of these chemical reacting flows will be a new tool that is needed in the design of these future propulsion concepts. Experimental and code development research is being performed at Lewis to better understand chemical reacting flows with the long term goal of establishing these reliable computer codes.

Our approach to understanding chemical reacting flows is to look at separate more simple parts of this complex phenomena as well as to study the full turbulent reacting flow process. As a result we are engaged in research on the fluid mechanics associated with chemical reacting flows. We are also studying the chemistry of fuel-air combustion. Finally, we are investigating the phenomena of turbulence-combustion interaction. This presentation will highlight research, both experimental and analytical, in each of these three major areas.

Chemical reacting flows of aerospace propulsion systems have features similar to other internal flows described previously. The flows are typically highly turbulent, with large secondary flows and three-dimensional flow characteristics. Flow oscillations and unsteadiness is usually noticed in these flows. There are additional features which are unique to flows with combustion which adds a great deal of complexity to the process. This includes a substantial increase in temperature as the flow moves downstream, and a significant change in fluid properties due to fluid species changes. In addition, the time scale for combustion is often orders of magnitude different than the fluid flow time, and there is often a strong interaction between the turbulent flow and the combustion process. These complex features not only make experimental studies difficult but also significantly affect the methods for computing these flows with computer codes.

CHEMICAL REACTING FLOWS

HAS FEATURES OF INTERNAL FLOWS:

- **TURBULENT**
- **HIGHLY THREE-DIMENSIONAL**
- **FLOW UNSTEADINESS**

HAS ADDITIONAL COMPLEXITY:

- **SUBSTANTIAL HEAT RELEASE**
- **MANY CHANGE OF SPECIES**
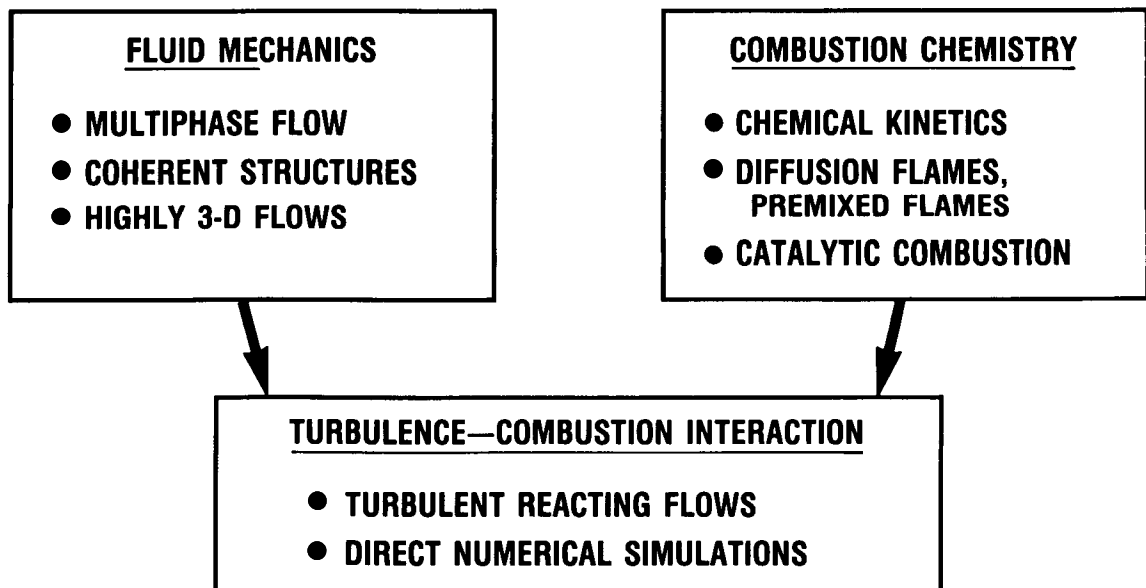- **COMBUSTION TIME** $<<$ **FLOW TIME SCALE**
- **FLOW—COMBUSTION INTERACTION**

CD–87–28754

The research activities in chemical reacting flows are divided into three areas as follows: (1) Fluid Mechanics, which looks at the fluid flow phenomena associated with combustion without the added complexity of including heat release. This includes the multiphase processes of fuel sprays, and the highly three-dimensional and time varying flows that typically exist in real propulsion systems. (2) Combustion Chemistry, which concentrates on the combustion of fuel and oxidizer without including the fluid mechanics. Research is being done to understand the ignition process of fuel and oxidizer and to probe the detailed chemistry to obtain an accurate combustion model for further fluid codes. Catalytic combustion is also being studied as a fuel processor for high speed propulsion. (3) Turbulence Combustion Interaction, which looks at both the fluid mechanics and the chemistry of combustion and their effects on each other. Work is being done to understand the key features of turbulent reacting flow and to construct accurate computer codes to simulate this flow. As a useful "numerical experiment," the techinque of direct numerical simulation is also being used to better understand chemical reacting flows.

These three major areas of activities are all being performed to achieve the long term goal to obtain an accurate predictive code with coupled fluid mechanics and chemistry which will be helpful in the design of future aerospace propulsion systems.

Let us now look at an example of the research in the area of Fluid Mechanics research on multiphase flow of liquid fuel sprays in air.

CHEMICAL REACTING FLOWS

FLUID MECHANICS

- MULTIPHASE FLOW
- COHERENT STRUCTURES
- HIGHLY 3-D FLOWS

COMBUSTION CHEMISTRY

- CHEMICAL KINETICS
- DIFFUSION FLAMES, PREMIXED FLAMES
- CATALYTIC COMBUSTION

TURBULENCE—COMBUSTION INTERACTION

- TURBULENT REACTING FLOWS
- DIRECT NUMERICAL SIMULATIONS

LONG TERM GOAL: ACCURATE PREDICTIVE CODE WITH COUPLED FLUID MECHANICS AND CHEMISTRY FOR FUTURE AEROSPACE PROPULSION
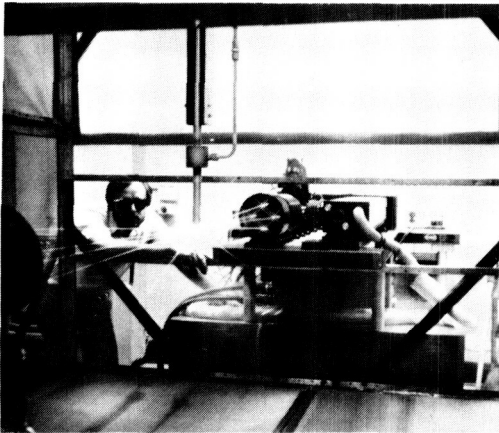
CD-87-28755

3-55

The fuel-spray process is extremely important in terms of engine efficiency, durability, and operability. The ultimate objective of the research is to develop a computer code that can accurately model the fuel and air mixing with subsequent combustion process. Since this is a very complicated process, it is being approached in a series of steps of increasing complexity. Particle-laden jets were initially studied in order to assess the capability of current two-phase flow models. Evaporating liquid sprays and then combusting sprays will also be studied.
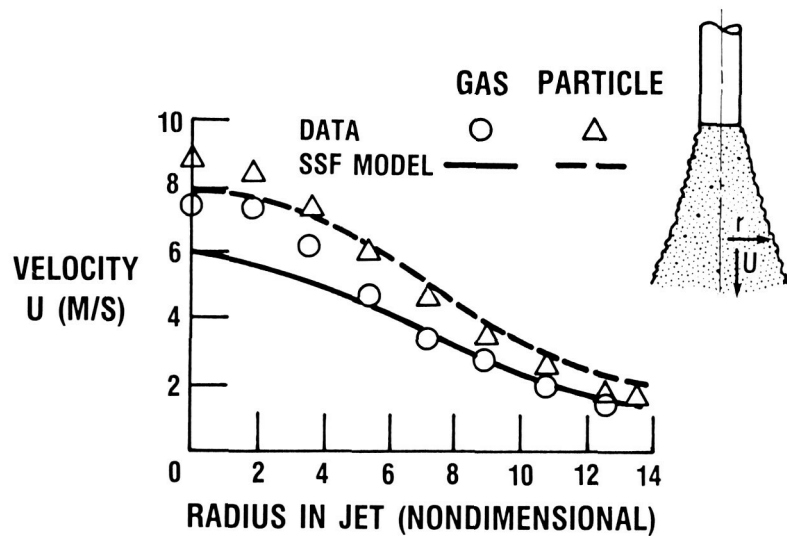
MULTIPHASE FLOWS

OBJECTIVE:

DEVELOP COMPUTER CODE WHICH ACCURATELY MODELS FUEL SPRAY/AIR MIXING AND COMBUSTION PROCESSES

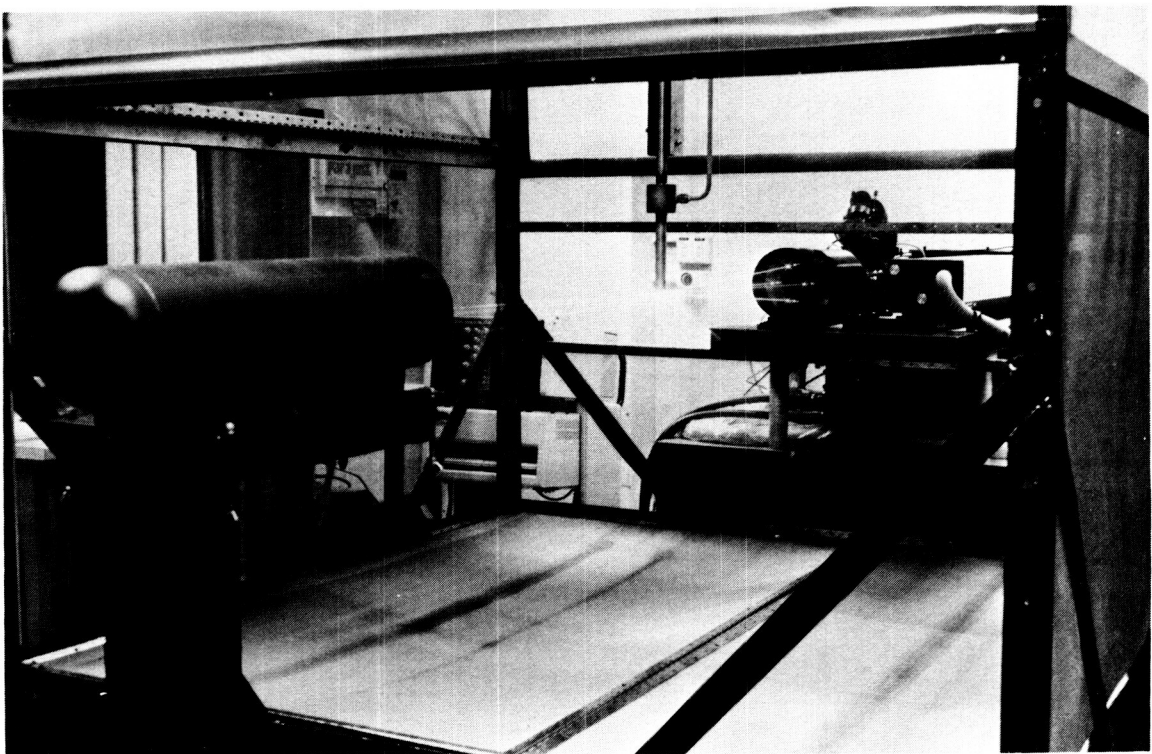**APPROACH:**

(1) PARTICLE-LADEN JET EXPERIMENT

(2) TWO-PHASE FLOW COMPUTER MODELS

CD-87-28756

PARTICLE-LADEN SWIRLING FLOW EXPERIMENTAL STUDY

This photograph illustrates the experimental arrangement of the particle-laden jet. An air jet containing solid glass beads (39 micron, Sauter Mean Diameter) discharged downward into a still environment. Particle-laden jets with three swirl numbers were studied. Nonintrusive measurements of velocity were obtained with a two-channel laser velocimeter. Particle size and velocity were measured with a phase/doppler particle anemometer. The gas phase was seeded with nominal 1-micron diameter aluminum oxide power to measure gas phase velocities.
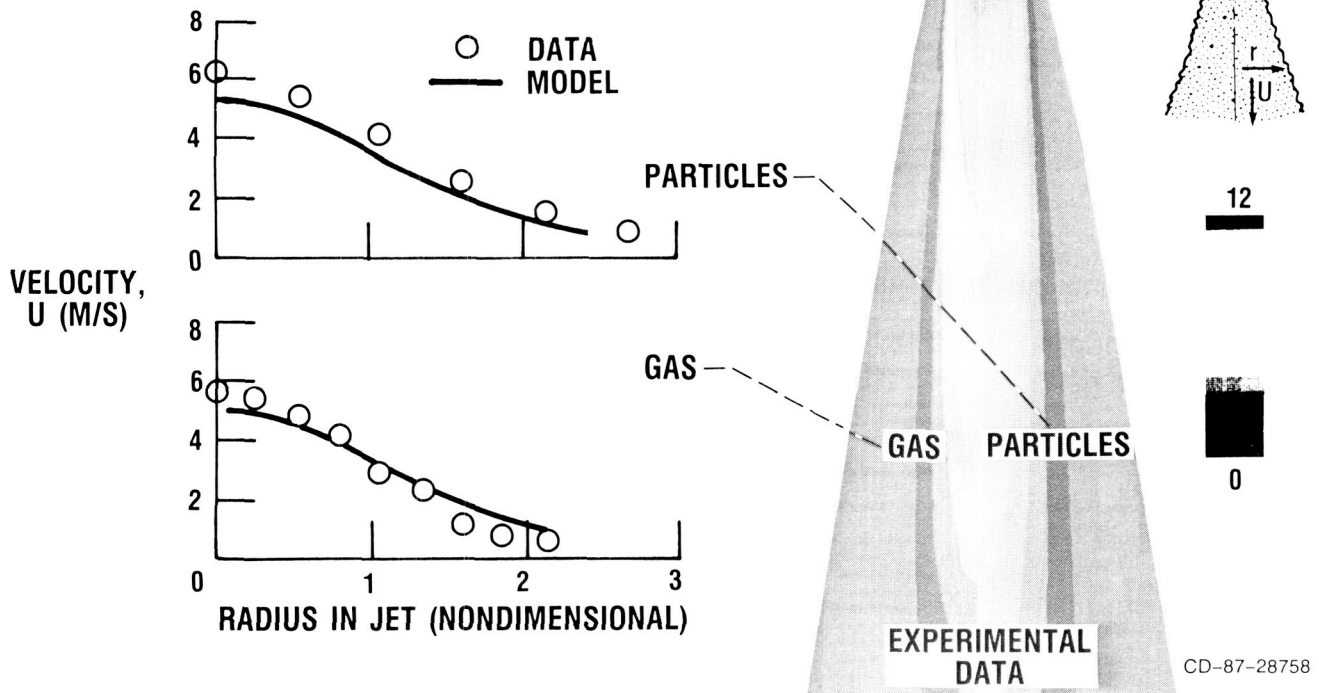
PARTICLE-LADEN SWIRLING FLOW EXPERIMENTAL STUDY

CD-87-28757

This figure presents typical results from the particle-laden jet study. A contour plot of experimentally measured axial velocity of the gas phase (left side) and particle phase (right side) is illustrated. It is evident that initially, the gas phase has a higher velocity than the particle phase. The particles are initially accelerated by the gas phase and then their velocity begins to decay. Because of their inertia, the rate of decay of axial velocity is slower for the particles than the gas. Also shown in the figure are predictions from the SSF model at 10 diameters downstream of the tube exit. This model tracks particle trajectories in the computed gas phase flowfield and allows momentum exchange between phases. The model also considers effects of gas-phase turbulence on particle trajectories. Predictions from the model show reasonable agreement with the data.

MULTIPHASE FLOW VELOCITY COMPARISON

CD-87-28758

Future directions for multiphase flow research include evaporating liquid sprays and combusting liquid sprays. Evaporating sprays are currently being studied under contract at the University of California, Irvine, and Allison Gas Turbine as part of the HOST Program. The test cell where the particle-laden jets were studied is currently being modified to study liquid sprays.
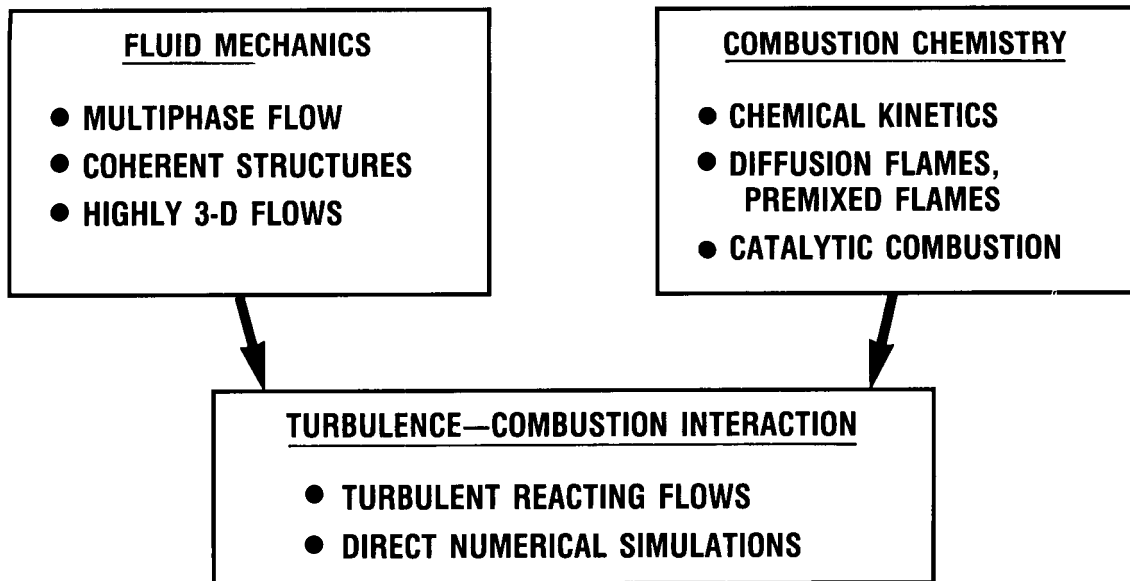
MULTIPHASE FLOWS

FUTURE DIRECTIONS:

(1) EVAPORATING LIQUID SPRAY
(2) LIQUID FUEL SPRAY COMBUSTION

Let us now turn our attention to the area of Combustion Chemistry. An example of the research in this area is the study of the chemical kinetics of hydrogen-air combustion.

CHEMICAL REACTING FLOWS

FLUID MECHANICS

- MULTIPHASE FLOW
- COHERENT STRUCTURES
- HIGHLY 3-D FLOWS

COMBUSTION CHEMISTRY

- CHEMICAL KINETICS
- DIFFUSION FLAMES, PREMIXED FLAMES
- CATALYTIC COMBUSTION

TURBULENCE—COMBUSTION INTERACTION

- TURBULENT REACTING FLOWS
- DIRECT NUMERICAL SIMULATIONS

LONG TERM GOAL: ACCURATE PREDICTIVE CODE WITH COUPLED FLUID MECHANICS AND CHEMISTRY FOR FUTURE AEROSPACE PROPULSION

CD-87-28755

CHEMICAL KINETICS

In a supersonic ramjet propulsion system, the time required between the injection of
fuel into the airstream and its combustion point is very important with regards to
the length of the engine and its weight. These high speed combustion concepts will
be tested in wind tunnels where the air has been heated to simulate the aerodynamic
heating of the vehicle in the atmosphere. Research is underway to determine the
combustion delay time of hydrogen fuel and air and to determine the effects of air
contaminants in the wind tunnel on this combustion delay time.

Stoichiometric H_2-O_2 ignition delay times were measured behind reflected shock waves
at 1.1 atm pressure over the temperature range 1300 to 950 K by using a chemical
shock tube. The proposed chemical kinetic model predicted ignition delay times in
excellent agreement with the experimental data.

CHEMICAL KINETICS

OBJECTIVE:
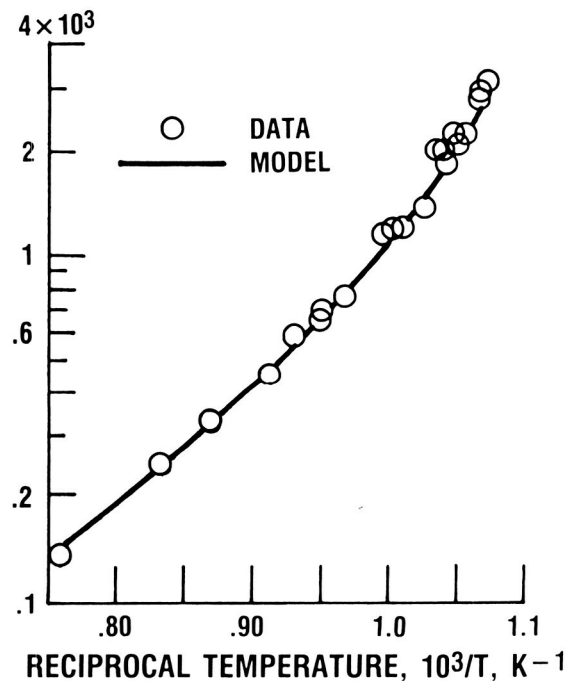
DETERMINE COMBUSTION DELAY TIME OF
HYDROGEN FUEL—AIR CHEMISTRY

APPROACH:
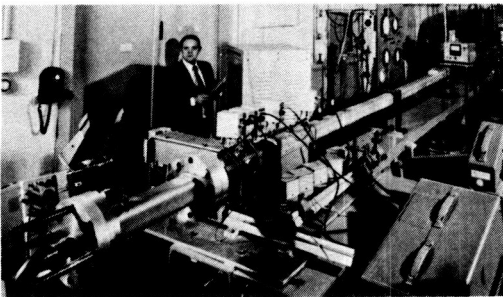
(1) CHEMICAL SHOCK TUBE

CD-87-28760

(2) REACTION RATES COMPUTER MODEL

The information which we are seeking is to determine how far from the fuel injection point will a stable flame exist in a supersonic airstream of various Mach numbers.

Also, since heated wind tunnels have small amounts of contaminants or additives in the air, we are studying the effects of these contaminants on this combustion delay time.

CHEMICAL KINETICS

SCRAMJET ENGINE SCHEMATIC:
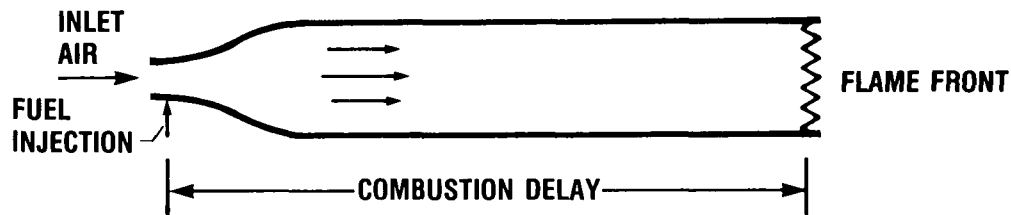
- **HOW FAR WILL STABLE FLAME BE FROM FUEL INJECTION POINT?**

- **WHAT IS EFFECT OF AIR CONTAMINANTS OR ADDITIVES ON THIS COMBUSTION DELAY?**

CD-87-28761

Shown here are the levels of four major air contaminants which exist in the two U.S. hypersonic wind tunnels when they are simulating a Mach 7 flight speed. Carbon dioxide and water vapor are known to lengthen the combustion delay time. Nitric oxide and nitrous oxide although orders of magnitude smaller in concentrations, would tend to shorten the combustion delay time.

CHEMICAL KINETICS

HEATED WIND TUNNEL AIR CONTAMINANTS

CARBON DIOXIDE .... 6 TO 10 PERCENT

WATER VAPOR .... 13 TO 17 PERCENT

NITRIC OXIDE .... 0.9 PERCENT

NITROUS OXIDE .... 0.025 PERCENT

CD-87-287762

Applying the chemical kinetics computer model to the prediction of combustion delay time for a scramjet flying at a Mach 7 flight condition predicted a 70 cm distance between the fuel injection point and the flame front. When the wind tunnel contaminants were taken into account, this distance was significantly shorter: only 13 cm between the fuel injection point and the flame front. Thus, the small concentration of contaminants (nitric oxide and nitrous oxide) resulted in over a factor of five reduction in combustion length. Not only is this effect important in evaluating wind tunnel experiments of combustion concepts, but it also indicates that trace additives into the flow could significantly shorten the required engine length and thereby considerably reduce weight.

This work is continuing to further explore the effects of these trace contaminants or additives and to better quantify their potential benefit to future scramjet designs.

CHEMICAL KINETICS

CALCULATED COMBUSTION DELAY FOR MACH 7 SCRAMJET FLIGHT CONDITION:

INLET AIR

FUEL INJECTION

FLAME FRONT

├──────── 70 cm ────────┤
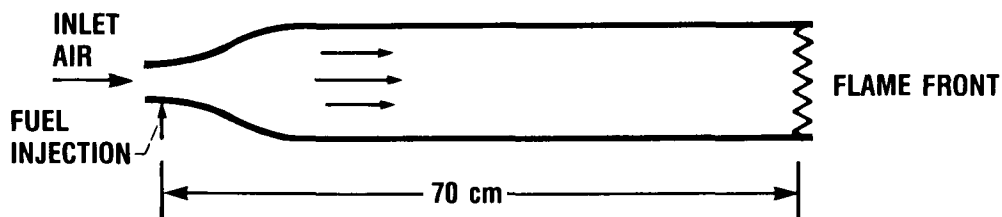
(A) BASED ON PURE DRY AIR.

INLET AIR

FLAME FRONT

FUEL INJECTION

|←13 cm→|
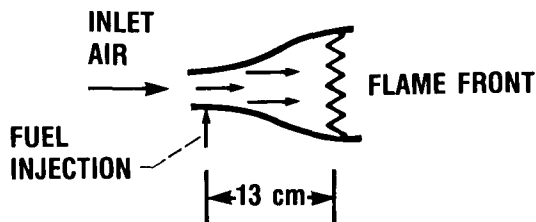
(B) BASED ON PURE AIR WITH CONTAMINANTS OR ADDITIVES.

CD-87-28763

While it is important and quite useful to look at the fluid mechanics and the combustion chemistry aspects of chemical reacting flows independently, to get a full understanding of the dominant phenomena of these flows, we must examine the interaction of tur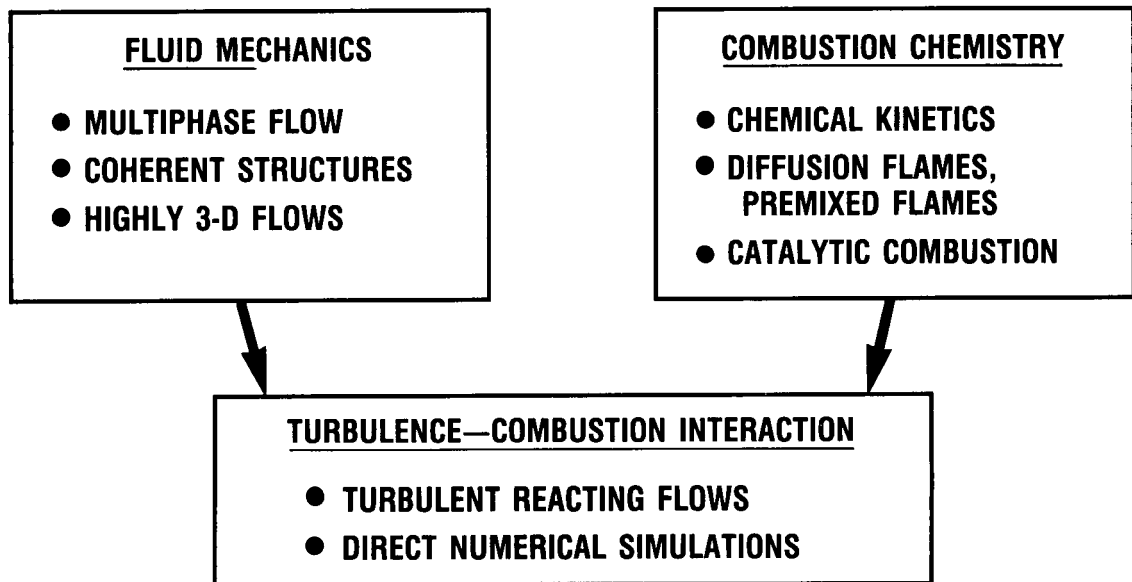bulent flow with combustion. The activities in this area include both numerical code development work and experimental research. We will now highlight the research in turbulent reacting flow.

CHEMICAL REACTING FLOWS

FLUID MECHANICS	COMBUSTION CHEMISTRY
● MULTIPHASE FLOW ● COHERENT STRUCTURES ● HIGHLY 3-D FLOWS	● CHEMICAL KINETICS ● DIFFUSION FLAMES, PREMIXED FLAMES ● CATALYTIC COMBUSTION

TURBULENCE—COMBUSTION INTERACTION

● TURBULENT REACTING FLOWS
● DIRECT NUMERICAL SIMULATIONS

LONG TERM GOAL: ACCURATE PREDICTIVE CODE WITH COUPLED FLUID MECHANICS AND CHEMISTRY FOR FUTURE AEROSPACE PROPULSION
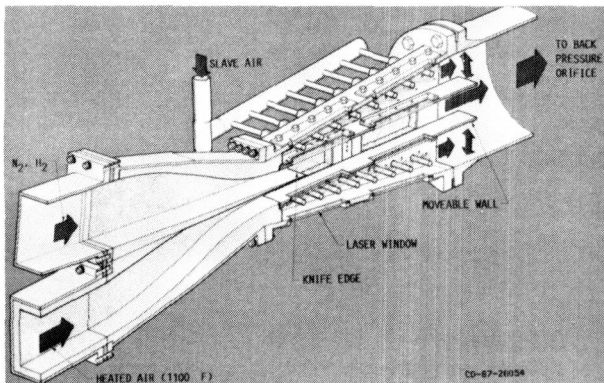
CD-87-28755

The objective of this work is to understand the coupling between fluid dynamics and combustion and to establish an accurate computer code which simulates the dominant features of turbulent reacting flow. An experiment is being constructed to focus on many of these features, both steady state and unsteady. This experiment is to examine a plane reacting free shear layer. Turbulent reacting flow computer codes, both steady state and time accurate, are also being developed concurrently, and the database from the experiment will serve as a means to validate these new computer codes.
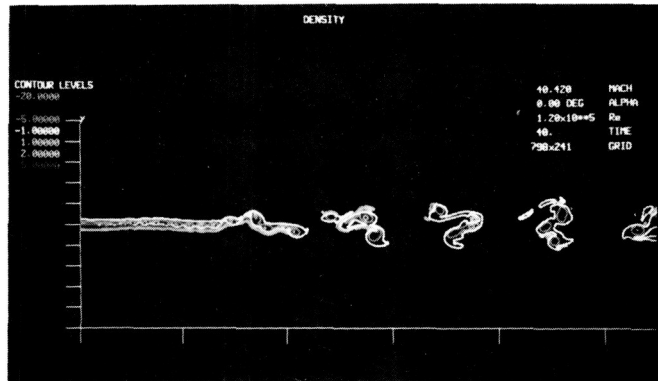
TURBULENT REACTING FLOW

OBJECTIVE:

UNDERSTAND COUPLING BETWEEN FLUID DYNAMICS AND COMBUSTION, AND ESTABLISH ACCURATE COMPUTER SIMULATIONS OF HIGH-SPEED CHEMICAL REACTING FLOW
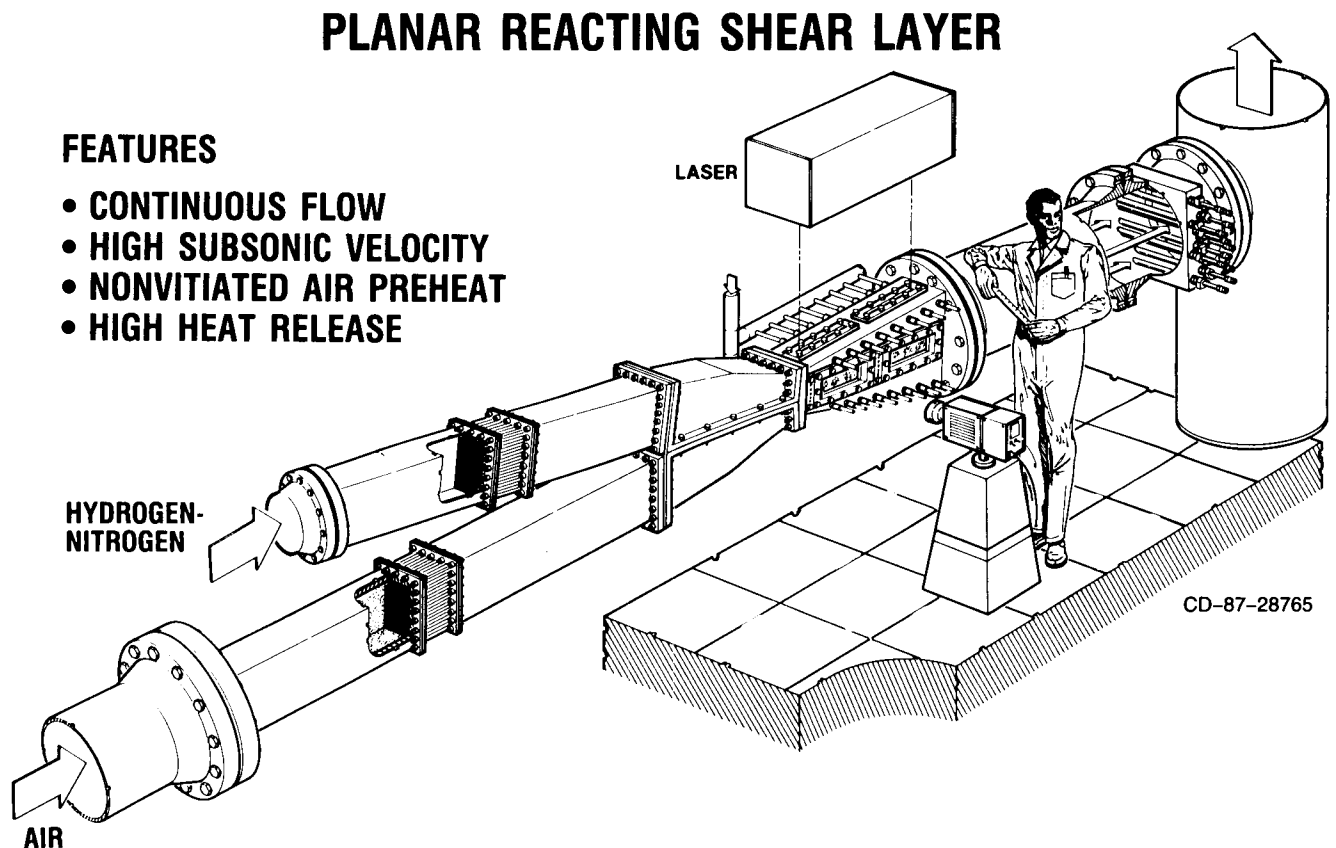
APPROACH:

(1) REACTING SHEAR LAYER EXPERIMENTS

(2) TURBULENT REACTING FLOW COMPUTER CODES

CD-87-28764

The objectives of the Planar Reacting Shear Layer Experiment, shown as a schematic here, is to (1) understand the coupling between fluid dynamics and combustion and (2) to establish a data set to validate computer codes which simulate the physics and chemistry of high speed chemical reacting flow. Two gas streams, one of hydrogen and nitrogen, the other of air, will mix downstream of a plane splitter plate. Combustion will occur where the fuel and air has properly mixed. Pressure oscillations in this closed duct will exist due to the dynamic features of the flow and the interactions between these pressure oscillations and the combusting shear layer will be examined. The unique features of this experiment are (1) a continuous flow capability, (2) flow velocities of both the air and the hydrogen-nitrogen mixture which are in the high subsonic range, (3) the air will be heated ahead of the mixing shear layer without any contamination effects, and (4) the heat release in this experiment will be quite high, typical of propulsion systems. The experiment is in fabrication and nonintrusive instrumentation is being purchased. Experiments are expected to begin in spring 1988.

PLANAR REACTING SHEAR LAYER

FEATURES
- CONTINUOUS FLOW
- HIGH SUBSONIC VELOCITY
- NONVITIATED AIR PREHEAT
- HIGH HEAT RELEASE

LASER

HYDROGEN-NITROGEN

AIR

CD-87-28765

PROMETHEUS II is a time-accurate version of a two-dimensional, finite volume TEACH code. Second-order accurate QUICK differencing is used for the convective terms and block-correction combined with Stone's strongly implicit algorithm is used to solve the pressure correction equation. This results in a highly efficient computional tool for performing "numerical experiments."

TURBULENT REACTING FLOW

TIME-ACCURATE, 2-D SHEAR LAYER CODE

- **TWO-DIMENSIONAL NAVIER-STOKES EQUATIONS SOLVED WITH OR WITHOUT A TWO-EQUATION TURBULENCE MODEL**

- **SECOND-ORDER ACCURACY IN BOTH TIME AND SPACE**

- **FULLY IMPLICIT NUMERICAL SCHEME DERIVED FROM "SIMPLE" (SEMI-IMPLICIT PRESSURE LINKED EQUATIONS) ALGORITHM**
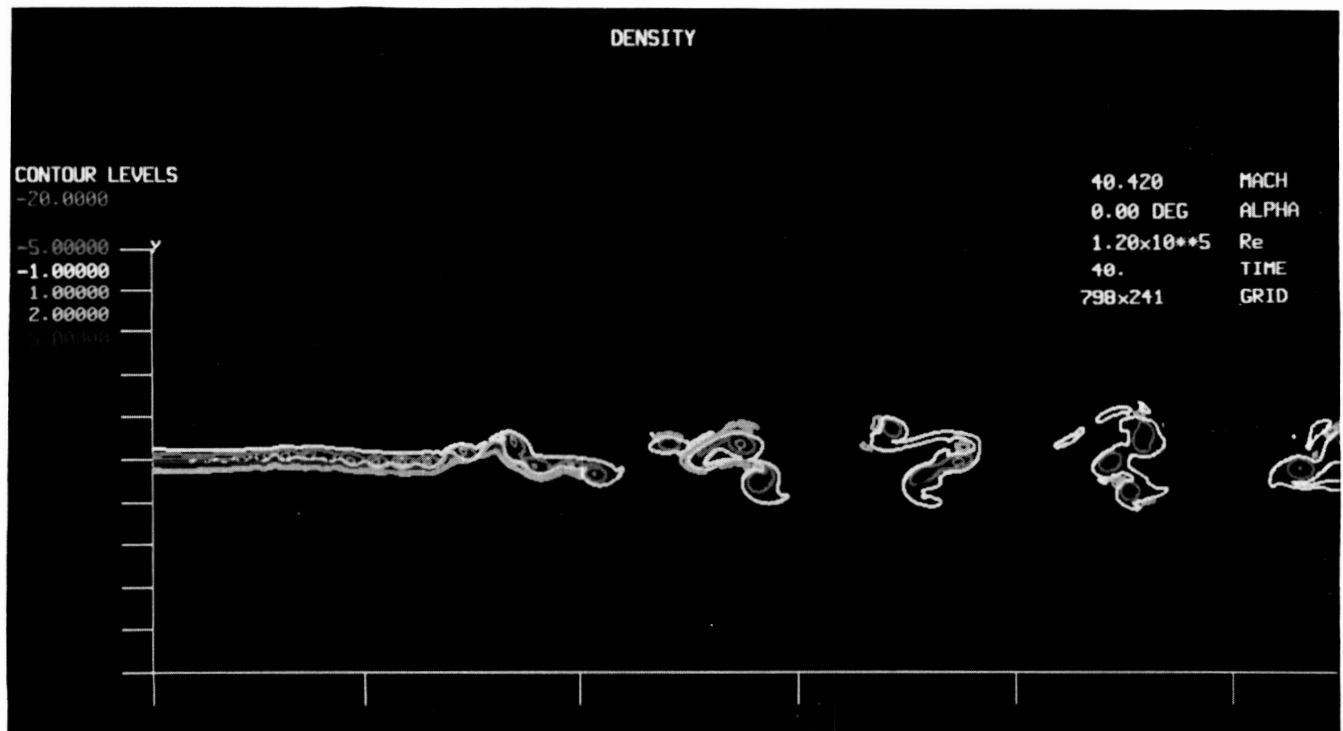
CD 87 28766

TURBULENT REACTING FLOW

Shown are vorticity contours for a two-dimensional, numerical calculation of a
forced, shear layer at a Reynolds number of about 100,000. The positive and
negative vorticity contours originate at the boundary layers, specified at the inlet
of the computational domain. Forcing is applied at a long wavelength and smaller
scale vorticities spontaneously develop as a result of the natural instability of
the layer. These small scale vorticities cluster on the scale of the longer, forced
wavelength. Small pockets of positive vorticity persist as remnants of the low
speed boundary layer. The collective interaction of these small scale vortices,
merging into larger scale structures, largely controls the dynamics of the shear
layer. These calculations were performed on the NAS Cray 2 computer.

TURBULENT REACTING FLOW

TIME-ACCURATE, 2-D SHEAR LAYER CODE

COMPUTER CODE RESULTS OF TWO STREAM FREE SHEAR LAYER SHOWING VORTICITY STRUCTURE

DENSITY

CONTOUR LEVELS
-20.0000

-5.00000 ──y
-1.00000
1.00000
2.00000

40.420   MACH
0.00 DEG ALPHA
1.20x10**5 Re
40.   TIME
798x241   GRID

The RPLUS2D code is being developed for calculation of mixing and chemical reactions in the flow fields of ramjets and scramjets. The code is written in generalized curvilinear coordinates and therefore can handle any arbitrary two-dimensional geometry. The implicit LU (lower-upper) scheme used in the RPLUS2D code requires only scalar diagonal inversions while most other implicit schemes require block matrix inversions. The use of scalar diagonal inversions offers order-of-magnitude efficiency improvement over conventional implicit schemes when large systems of partial differential equations must be solved, such as flows in ramjet and scramjet engines. A three-dimensional version of this code will be developed subsequent to the completion of this two-dimensional code.

TURBULENT REACTING FLOW

STEADY STATE FLUID MECHANICS COMPUTER CODE:

IMPLICIT L U SYMMETRIC GAUSS-SEIDEL ALGORITHM WITH FINITE VOLUME DISCRETIZATION

- **MAJOR ADVANTAGE OF ELIMINATING BLOCK INVERSIONS WHERE THE NUMBER OF BLOCKS COULD BE QUITE LARGE FOR REACTING FLOWS**
- **REAL GAS PROPERTIES**
- **KINETIC REACTION RATE SOURCE TERMS TREATED IMPLICITLY**

CD-87-28000

The H_2 jet injected at right angles to the main airflow acts essentially as an effective body which blocks the flow resulting in a bow shock and strong pressure gradients on the front side of the jet. The jet shock around the injector can also be seen in this figure. The two bow shocks resulting from the two H_2 jets at top and bottom walls (symmetric about the centerline shown) intersect at the center plane and yield a large pressure rise.

TURBULENT REACTING FLOW

STEADY STATE FLUID MECHANICS COMPUTER CODE:

COMPUTER CODE RESULTS OF GASEOUS FUEL JET—AIR MIXING AND COMBUSTION IN INTERNAL DUCT FLOW, SHOWING STATIC PRESSURE CONTOURS
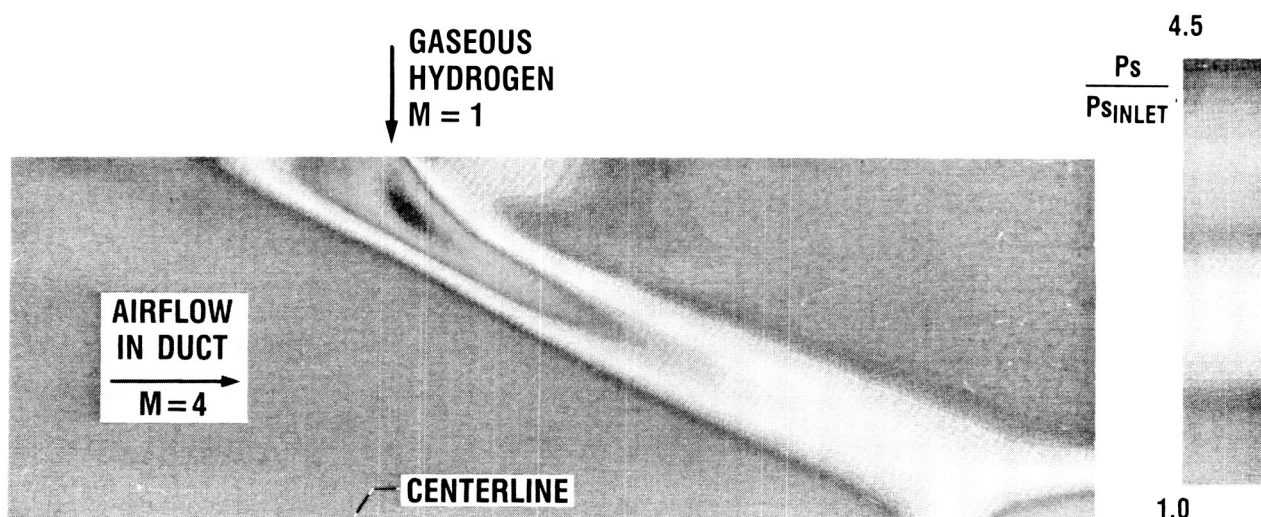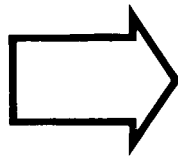

CD-87-28769

In conclusion, research in the area of Chemical Reacting Flow will lead to an understanding of this complex flow and accurate predictive computer codes. Activity in this topic is focused on three areas: Fluid Mechanics, Combustion Chemistry, and Turbulence-Combustion Interaction. Results from this research will provide important technical "tools we need" to develop new aerospace propulsion systems for the year 2000 and beyond.

CHEMICAL REACTING FLOWS

TOOLS WE NEED

FLUID MECHANICS

COMBUSTION CHEMISTRY

TURBULENCE—COMBUSTION
INTERACTION

NEW
AEROSPACE
PROPULSION

CD-87-28770

NASA
National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA CP-10003	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Aeropropulsion '87 Session 3 - Internal Fluid Mechanics Research		5. Report Date November 1987
		6. Performing Organization Code
7. Author(s)		8. Performing Organization Report No. E-3798
		10. Work Unit No. 505-62-3B
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		11. Contract or Grant No.
		13. Type of Report and Period Covered Conference Publication
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

Preprint includes figures and descriptive text.

16. Abstract

Internal fluid mechanics research at Lewis is directed toward an improved understanding of the important flow physics affecting aerospace propulsion systems, and applying this improved understanding to formulate accurate predictive codes. To this end, research is conducted involving detailed experimentation and analysis. The presentations in this session summarize ongoing work and indicate future emphasis in three major research thrusts; namely, inlets, ducts and nozzles, turbomachinery, and chemical reacting flows.

17. Key Words (Suggested by Author(s)) Aeronautical propulsion research; Materials; Structures; Internal fluid mechanics; Instrumentation; Controls; Subsonic propulsion; Supersonic propulsion; Hypersonic propulsion	18. Distribution Statement Unclassified - Unlimited Subject Category 07		
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No of pages 74	22. Price* A04

NASA FORM 1626 OCT 86 *For sale by the National Technical Information Service, Springfield, Virginia 22161